

NASA/CR—2000-209308



# CSTEM User Manual

M. Hartle and R.L. McKnight  
General Electric Aircraft Engines, Cincinnati, Ohio

---

January 2000

## The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to [help@sti.nasa.gov](mailto:help@sti.nasa.gov)
- Fax your question to the NASA Access Help Desk at (301) 621-0134
- Telephone the NASA Access Help Desk at (301) 621-0390
- Write to:  
NASA Access Help Desk  
NASA Center for Aerospace Information  
7121 Standard Drive  
Hanover, MD 21076

NASA/CR—2000-209308



# CSTEM User Manual

M. Hartle and R.L. McKnight  
General Electric Aircraft Engines, Cincinnati, Ohio

Prepared under Contract NAS3-26617

National Aeronautics and  
Space Administration

Glenn Research Center

---

January 2000

Available from

NASA Center for Aerospace Information  
7121 Standard Drive  
Hanover, MD 21076  
Price Code: A20

National Technical Information Service  
5285 Port Royal Road  
Springfield, VA 22100  
Price Code: A20



## TABLE OF CONTENTS

<b>ABOUT THE CSTEM USER MANUAL .....</b>	<b>x</b>
<b>SECTION 1. CSTEM OVERVIEW .....</b>	<b>1.1</b>
<b>SECTION 2. MODELING CONSIDERATIONS .....</b>	<b>2.1</b>
2.1. COORDINATE SYSTEMS .....	2.1
2.1.1. Global Coordinate System (XYZ) .....	2.1
2.1.2. Skew Coordinate Systems (xyz) .....	2.1
2.1.3. Structural Coordinate System (rst) .....	2.1
2.1.4. Material Coordinate System (123) .....	2.2
2.1.5. Micromechanics Coordinate System (xyz) .....	2.3
2.1.6. Mesh Generation Coordinate System ( $x''y''z''$ ) .....	2.4
2.1.7. Wave Coordinate System (xyz) .....	2.4
2.2. SIGN CONVENTIONS .....	2.5
2.2.1. Angles .....	2.5
2.2.2. Forces/Displacements .....	2.5
2.2.3. Pressures .....	2.5
2.2.4. Stress/Strain .....	2.6
2.3. TRANSFORMATIONS .....	2.6
2.3.1. Coordinate System Transformations .....	2.6
2.3.1.1 Coordinate Transformation .....	2.6
2.3.1.2 Stress/Strain Coordinate System Transformations .....	2.7
2.3.1.3 Material Coordinate System Transformations .....	2.8
2.3.1.4 Coordinate System Permutations .....	2.9
2.3.2. Reference Configuration Transformations .....	2.9
2.3.2.1 Deformation Gradient .....	2.9
2.3.2.2 Density Transformation .....	2.10
2.3.2.3 Strain Transformation .....	2.10
2.3.2.4 Stress Transformation .....	2.11
2.3.2.5 Material Transformation .....	2.11
2.4. ELEMENTS .....	2.11
2.4.1. 3D Elements .....	2.11
2.4.2. Equivalent Homogeneous Elements .....	2.17
2.4.3. Layered Elements .....	2.18
2.4.4. Additional Stiffnesses .....	2.26
2.5. CONSTRAINT EQUATIONS .....	2.26
<b>SECTION 3. ANALYSIS TYPES .....</b>	<b>3.1</b>
3.1. STRUCTURAL ANALYSES .....	3.1
3.1. 1. Load Types .....	3.1
3.1. 2. Material Properties .....	3.2
3.1. 2. 1. Direct Input of Material Properties .....	3.3
3.1. 2. 2. Calculation of Material Properties .....	3.3
3.1. 3. Small Displacement Analysis .....	3.3
3.1. 3. 1. Stress Recovery .....	3.4
3.1. 3. 1.1. Failure Criteria .....	3.4
3.1. 4. Large Displacement Analysis .....	3.5

3.1. 5. Finite Element Implementation of Large Displacement Analysis .....	3.8
3.1. 5. 1. Deformation Gradient .....	3.9
3.1. 5. 2. Stiffness Calculations .....	3.9
3.1. 5. 3. Force Calculations and Assembly .....	3.10
3.1. 5. 4. Equilibrium Iterations .....	3.10
3.1. 5. 5. Stress Recovery .....	3.11
3.1. 6. Principal Stress Calculations .....	3.11
3.1. 7. Free Vibration Analysis .....	3.12
3.1. 8. Buckling Analysis .....	3.15
3.1. 9. Micromechanics Analysis .....	3.16
3.1. 9. 1. Data Bank of Constituent Properties .....	3.16
3.1. 9. 2. Calculation of Ply Material Properties .....	3.16
3.1. 9. 3. Detailed Microanalyses .....	3.17
3.1. 10. Enhanced Shear Stress Recovery .....	3.22
3.1. 11. Nonlinear Material Analysis .....	3.25
3.2. HEAT TRANSFER ANALYSIS .....	3.47
3.2. 1. Introduction .....	3.47
3.2. 2. Finite Element Implementation .....	3.47
3.2. 3. Equations for Heat Transfer Analysis .....	3.49
3.2. 4. Boundary Conditions .....	3.50
3.2. 5. Solution Scheme .....	3.51
3.2. 6. Graded Composite Structures .....	3.56
3.3. ELECTROMAGNETIC ANALYSIS .....	3.59
3.3. 1. Data Bank .....	3.60
3.3. 2. Exposure Analysis .....	3.60
3.3. 3. Wave Matrix Method .....	3.61
3.3. 4. Optics Method .....	3.61
3.3. 5. Table Look Up Method .....	3.65
3.4. ACOUSTICS ANALYSIS .....	3.67
3.4. 1. Theoretical Approach .....	3.67
3.4. 2. Finite Element Implementation .....	3.70
3.4. 3. Acoustics References .....	3.72
3.5. TAILORING METHODOLOGY .....	3.73

<b>SECTION 4. DESCRIPTION OF INPUT .....</b>	<b>4.1</b>
4.1. ANALYSIS CONTROL AND STRUCTURAL INPUT .....	4.3
4.1.1. ANALYSIS CONTROL .....	4.3
4.1.1.1. Initial Control Lines .....	4.3
4.1.1.1.1. Title Line .....	4.3
4.1.1.1.2. Problem Size Line .....	4.4
4.1.1.1.3. Restart and Restart File Creation Options .....	4.6
4.1.1.1.4. Analysis Options Line .....	4.7
4.1.1.2. Load Case Control Lines .....	4.9
4.1.1.2.1. First Load Case Control Line .....	4.9
4.1.1.2.2. Second Load Case Control Line .....	4.12
4.1.1.3. Keyword Input .....	4.15
4.1.1.3.1. Debug Printout Controls, PDBG, DDBG, and CDBG .....	4.15
4.1.1.3.2. Eigenanalysis Solution Parameters, EIGS .....	4.16
4.1.1.3.3. Large Displacement Buckling, LDBK .....	4.16
4.1.1.3.4. Nonlinear Force Calculation Method, METH .....	4.16

4.1.1.3.5. Newton–Raphson Iteration Method, NRIT .....	4.16
4.1.1.3.6. Damage Acceleration, DACC .....	4.17
4.1.1.3.7. Damage Iteration Averaging, DAVG .....	4.17
4.1.1.3.8. Read Alternate Stress Stiffness, RDKS .....	4.18
4.1.1.3.9. Change Vibration Boundary Conditions, VIBC .....	4.18
4.1.1.3.10. Microstress Plot Output, MICR .....	4.18
4.1.1.3.11. Material Selected Stress Summary, MATX .....	4.19
4.1.1.3.12. Linear Load Ramping, RAMP .....	4.19
4.1.1.4. File SLOP Mode Shape Slope Input .....	4.20
4.1.2. GEOMETRY .....	4.20
4.1.2.1. Nodes .....	4.20
4.1.2.2. Elements .....	4.21
4.1.2.3. Mesh Generation .....	4.22
4.1.2.4. Node / Element Examples .....	4.23
4.1.2.4.1. Basic Input Example .....	4.23
4.1.2.4.2. Mesh Generation Input Example .....	4.26
4.1.3. ELASTIC MATERIAL PROPERTIES .....	4.28
4.1.4. LAYERING .....	4.29
4.1.4.1. Manual Layering Method .....	4.30
4.1.4.2. Cross Section Layering With Assignment To Multiple Cross Sections .....	4.33
4.1.4.3. Cross Section Layering With Generation For Each Cross Section ...	4.35
4.1.4.4. Special Layering Considerations .....	4.35
4.1.4.5. Layer Orientation Visualization .....	4.42
4.1.5. CONSTRAINT EQUATIONS .....	4.42
4.1.6. ADDITIONAL STIFFNESSES .....	4.43
4.1.7. TRANSFORMATIONS .....	4.43
4.1.8. NODAL FIXITIES .....	4.44
4.1.9. NODAL DISPLACEMENTS AND FORCES .....	4.44
4.1.10. NODAL TEMPERATURES .....	4.44
4.1.11. PRESSURE LOADS .....	4.45
4.1.12. PLASTICITY ANALYSIS .....	4.45
4.1.12.1. Activating A Plasticity Analysis .....	4.45
4.1.12.2. Plasticity Iteration Control .....	4.46
4.1.12.3. Plasticity Material Properties .....	4.47
4.1.12.3.1. Isotropic Material Plasticity .....	4.47
4.1.12.3.2. C.T. Sun Orthotropic Plasticity .....	4.48
4.1.12.3.3. User Plasticity .....	4.49
4.1.13. CREEP ANALYSIS .....	4.50
4.1.13.1. Activating A Creep Analysis .....	4.50
4.1.13.2. Time Increment Control .....	4.51
4.1.13.3. Creep Iteration Control .....	4.52
4.1.13.4. Creep Material Properties .....	4.52
4.1.13.4.1. Isotropic Material Creep .....	4.53
4.1.13.4.2. C.T. Sun Orthotropic Creep .....	4.53
4.1.14. DAMAGE MECHANICS ANALYSIS .....	4.54
4.1.14.1. Activating A Damage Analysis .....	4.55
4.1.14.2. Damage Analysis Iteration Control .....	4.55
4.1.14.3. CMCUMAT Damage Mechanics .....	4.56
4.1.14.3.1. Polynomial Stiffness Degradation Model .....	4.57
4.2. HEAT TRANSFER ANALYSIS INPUT .....	4.59

4.2.1. Heat Transfer Analysis Types .....	4.59
4.2.2. Structural Coupling .....	4.59
4.2.3. Heat Transfer Load Case Control Information .....	4.59
4.2.3.1. Time Step and Temperature Points .....	4.60
4.2.3.2. Analysis Options .....	4.61
4.2.3.3. Restart .....	4.62
4.2.4. Material Properties .....	4.62
4.2.5. Data Sets .....	4.63
4.2.6. Internal Heat Generation .....	4.63
4.2.7. Convection Heat Transfer .....	4.64
4.2.8. Surface Heat Flux .....	4.65
4.2.9. Nodal Heat Flux .....	4.65
4.2.10. Prescribed Nodal Temperatures .....	4.66
4.2.11. Radiation Heat Transfer .....	4.66
4.3. ELECTROMAGNETIC ANALYSIS INPUT .....	4.68
4.3.1. Material Description .....	4.68
4.3.2. Electromagnetic Wave, Path, and Load Case Specifications .....	4.69
4.3.3. Data Bank Formats .....	4.70
4.4. ACOUSTICS ANALYSIS INPUT .....	4.72
4.4.1. Nodal Forcing Function .....	4.72
4.4.2. Modal Loss Factor .....	4.72
4.4.3. Masking Point .....	4.72
4.4.4. Results of Acoustic Analysis .....	4.73
4.5. TAILORING INPUT .....	4.74
4.5.1. Input Form .....	4.74
4.5.2. Procedure Type .....	4.74
4.5.3. Control Variables .....	4.75
4.5.4. Tolerances .....	4.75
4.5.5. Object and Design Variables .....	4.75
4.5.6. Constraints .....	4.76
4.5.7. Current Tailoring Procedures .....	4.76
4.5.8. Layered Tailoring .....	4.81
<b>SECTION 5. INPUT SHEETS .....</b>	<b>5.1</b>
5.1. ANALYSIS CONTROL AND STRUCTURAL INPUT .....	5.8
5.2. HEAT TRANSFER ANALYSIS INPUT .....	5.62
5.3. ELECTROMAGNETIC ANALYSIS INPUT AND DATA BANK .....	5.76
5.4. ACOUSTICS ANALYSIS INPUT .....	5.86
5.5. TAILORING CONTROL INPUT .....	5.89
<b>SECTION 6. DESCRIPTION OF OUTPUT .....</b>	<b>6.1</b>
6.1. INPUT ECHO, FILE1 (NT1) .....	6.2
6.2. STRUCTURAL RESULTS, FILE4 (NT4) .....	6.2
6.3. MODESHAPE STRESS/STRAIN OUTPUT, FILE46 (NT46) .....	6.3
6.4. STRUCTURAL RESTART, FILE10 (NT10) .....	6.3
6.5. ELEMENT STIFFNESS, FILE7 (NT7) .....	6.4
6.6. LAYER VISUALIZATION, FILE18 (NT18) .....	6.4
6.7. HEAT TRANSFER INPUT ECHO AND RESULTS, FILE31 (NT31) ..	6.4
6.8. HEAT TRANSFER RESTART, FILE37 (NT37) .....	6.5
6.9. ELECTROMAGNETIC INPUT ECHO, FILE51 (NT51) .....	6.5

6.10. ACOUSTICS RESULTS, FILE59 (NOSND) .....	6.5
6.11. DEBUG INFO, FILE60 (NTDBG) .....	6.5
6.12. ICAN RESULTS / INTERPOLATED STRESS, FILE61 (NT61) .....	6.5
6.13. INTERPOLATED CROSS SECTION STRAIN, FILE62 (NT62) .....	6.6
6.14. RAX VIBRATION RESULTS, FILE68 (NFPD) .....	6.6
6.15. SIESTA VIBRATION MODE SHAPES, FILE69 (NT69) .....	6.6
6.16. GENERAL TAILORING RESULTS, FILE76 (I6I) .....	6.7
6.17. SPECIFIC TAILORING RESULTS, FILE93 (93) .....	6.7
6.18. SCRATCH FILES .....	6.7
6.18.1. Element Stiffness and Integration Point Data, FILE7, FILE22, FILE42	6.7
6.18.2. Integration Point Constitutive Data, FILE8 .....	6.8
6.19. PATRAN RESULTS FILES .....	6.8
6.19.1. Displacement Results File, PATRANDISP .....	6.8
6.19.2. Nodal Temperature Results File, PATRANTEMP .....	6.8
6.19.3. Nodal Results File, PATNDRES .....	6.9
6.19.4. Element Stress Results File, PATELSIG .....	6.9
6.19.5. Element Coordinate System File, PATRANCRD .....	6.10
6.19.6. Modeshape Displacement Files, PATFREQ and PATBUCK .....	6.10
6.19.7. Damage Parameter File, PATDAMGP .....	6.10

## **APPENDIX 1. CSTEM FILE STRUCTURE**

## **APPENDIX 2. WAVES USER MANUAL**

## **APPENDIX 3. ICAN USER MANUAL**

## **APPENDIX 4. PATRAN PREFERENCE MANUAL**

## LIST OF FIGURES

FIGURE 1. CSTEM FLOWCHART .....	1.6
FIGURE 2. TAILORING PROCESS FLOWCHART .....	1.7
FIGURE 3. STRUCTURAL COORDINATE SYSTEM ORIENTATION FOR 3D ELEMENTS .....	2.2
FIGURE 4. MATERIAL COORDINATE SYSTEM FOR COMPOSITE LAMINATE .	2.3
FIGURE 5. WAVE COORDINATE SYSTEM FOR 0 DEG. POLARIZED EM WAVE	2.5
FIGURE 6. COORDINATE TRANSFORMATION MATRIX EXPRESSED AS DIRECTION COSINES .....	2.7
FIGURE 7. STRAIN TRANSFORMATION MATRIX .....	2.8
FIGURE 8. ACCURACY OF LAYERED INTEGRATION .....	2.19
FIGURE 9. LAYERS IN AN ELEMENT .....	2.22
FIGURE 10. FORMATION OF ORTHOGONAL MATERIAL COORDINATE SYSTEM .	2.24
FIGURE 16. 3D ISOPARAMETRIC ELEMENT (8, 16, or 20 NODES) .....	3.48
FIGURE 17. 3D SOLUTION DOMAIN FOR GENERAL HEAT CONDUCTION ...	3.49
FIGURE 18. LAYERED MODEL .....	3.57
FIGURE 19. ELECTROMAGNETIC WAVE AT MATERIAL INTERFACE .....	3.65
FIGURE 20. ILLUSTRATION OF ACOUSTIC PROBLEM .....	3.68
FIGURE 21. SINGLE ELEMENT CYLINDER SECTION .....	4.24
FIGURE 22. BASIC NODE AND ELEMENT INPUT .....	4.25
FIGURE 23. MESH GENERATION EXAMPLE .....	4.26
FIGURE 24. GENERATED DOUBLE CURVED SURFACE .....	4.27
FIGURE 25.a. COMPOSITE BEAM .....	4.31
FIGURE 25.b. COMPOSITE BEAM ELEMENT NUMBERING .....	4.31
FIGURE 25.c. ELEMENT FACE NUMBERS .....	4.31
FIGURE 26. MANUAL LAYERING EXAMPLE .....	4.32
FIGURE 27. LAYERING ASSIGNMENT TO MULTIPLE CROSS SECTIONS .....	4.33
FIGURE 28. LAYERING GENERATION FOR EACH CROSS SECTION .....	4.36
FIGURE 28.a. ORTHOGONAL STRUCTURAL COORDINATE SYSTEM .....	4.38
FIGURE 28.b. NON-ORTHOGONAL THROUGH THICKNESS AXIS .....	4.39
FIGURE 28.c. R AXIS CAN BE USED AS MATERIAL BASIS AXIS .....	4.40
FIGURE 29. CMCUMAT DAMAGE MATERIAL PROPERTY INPUTS .....	4.58
FIGURE 30. TEMPLATE FOR THE "ANALIZn" ROUTINE .....	4.78

## LIST OF TABLES

TABLE 1. THERMAL ANALYZER PARAMETERS .....	1.3
TABLE 2. GAUSS-LEGENDRE QUADRATURE POINTS AND WEIGHTS ..	2.17
TABLE 3. STRUCTURAL COORDINATE SYSTEM DEFINITION .....	2.21
TABLE 4. UNROTATED MATERIAL ORIENTATION WITH ELEMENT	2.23
TABLE 5. ELEMENT FACE NUMBERING CONVENTION .....	3.2
TABLE 6. TAILORING GLOBAL COMMON .....	4.79
TABLE 7. CURRENT TAILORING PROCEDURES .....	4.80

## ABOUT THE CSTEM USER MANUAL

This manual is a combination of a user manual, theory manual, and programmer manual. The reader is assumed to have some previous exposure to the finite element method. This manual is written with the idea that the CSTEM user needs to have a basic understanding of what the code is actually doing in order to properly use the code. For that reason, the underlying theory and methods used in the code are described to a basic level of detail.

Section 1 of this manual gives an overview of the CSTEM code: how the code came into existence, a basic description of what the code does, and the order in which it happens (a flowchart).

Section 2 starts off with a description of the various coordinate systems used, since they are of considerable importance to the proper use of the CSTEM code. The sign conventions, permutations, and transformations used in the code are described. The elements available in the CSTEM code are also described.

Section 3 describes the various types of analyses that can be performed with the CSTEM code with a description of what the code is doing in these analyses. This is more of a theory section.

Section 4 goes over the input. General input rules and procedures are described. Input variables and what they control are described in detail by function rather than by order of input.

Section 5 gives the order that the input variables are to be placed in the input decks. Each input deck for the different disciplines is described. A brief description of the input variables is given. Once a user becomes familiar with the CSTEM code, this is the section that will probably be referred to the most.

Section 6 describes what the output files are and what they contain.

Appendix 1 provides a listing and very brief description of every file used by the CSTEM code, including the type of file it is, what routine regularly accesses the file, and what routine opens the file.

Appendix 2 contains the WAVES user manual. WAVES is an electromagnetic analysis code contained within the CSTEM code.

Appendix 3 contains the ICAN user manual. ICAN is a micromechanics code contained within the CSTEM code.

Appendix 4 contains the PATRAN preference manual for CSTEM. The PATRAN preference is a separate combination of PATRAN PCL routines, FORTRAN executables, and UNIX scripts which allows the user to use PATRAN as a pre- and post-processor for the CSTEM finite element code.



## **SECTION 1. CSTEM OVERVIEW**

"CSTEM" is the acronym for the computer program developed under the NASA contract, "Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures". The technical objectives for this program are to produce radar signal transparent structures having high structural performance and low cost. The multidisciplines involved are all highly nonlinear. They include anisotropic, large deformation structural analysis, anisotropic thermal analysis, anisotropic electromagnetic analysis, acoustics, and coupled discipline tailoring. The CSTEM system is a computerized multidiscipline simulation specialized to the design problems of composite material radar absorbing structures. The enabling technical capabilities are implemented in a special 3D finite element formulated to simultaneously tailor the geometrical, material, loading, and environment complexities of radar transparent structures for cost effective optimum performance.

In each enabling technical discipline a decoupled stand-alone 3D finite element code has been developed. An executive program with controlling iterative solution techniques performs the nonlinear coupling among the participating analysis modules. Each analysis module is self contained, passing only the required input geometry and control information between the modules as well as returning any results which may be required as input for an analysis by a following module.

The structural module uses 8, 16, or 20 noded isoparametric bricks and is similar to many other isoparametric finite element codes in many ways. It has the capability for centrifugal, acceleration, nodal displacement, nodal force, temperature, and pressure loadings. An internal bandwidth optimizer is used to minimize problem size. The solution technique used is a multi-block column solver, which allows solution of very large problems since it can work on portions of the set of equations separately.

One of the more advanced features of the structural module is its orthotropic material capability. Material properties can be input relative to the material axes and then skewed on an element by element basis to obtain the desired orientation of the material with the global coordinate system. Material properties may also be referenced to the elemental coordinate system, with the orientation between elemental and global being calculated internally. The structural module can also generate the orthotropic material properties it needs for composite materials using the constituent properties making up the composite. This is done using an internal adapted version of the computer program INHYD, which accesses a data bank containing the material properties of the constituents. The properties are calculated based on the volume ratios of the constituents.

Another advanced feature of the CSTEM structural module is its multiple layer capability. This allows the modeling of composite structures with many material layers without the necessity of using an element for each layer. The stiffness of an element with multiple layers is calculated using integration points located on the midplane of each layer within the element. The composite stiffness gradient controls the finite element definition of a structure with two major parameters to vary: the

number of elements through the gradient and the number of numerical quadrature points within an element. A unique set of local stiffness characteristics is developed for each numerical integration point. Integration of these local characteristics over the volume of the element provides total element simulation of composite structures, including such effects as twist-bend coupling. The stress and strain are then recovered at these same integration points, with mapping options for results at other locations.

The structural analyzer also performs large deformation analysis using a unique incremental updated Lagrangian approach with iterative refinement. Testing of this capability against classical large deformation problems has shown it to be both more accurate and more economical than available alternatives. Connected with this capability is a deformed position eigen-analysis capability. All or selected portions of the nonlinear stiffness terms can be incorporated into these eigen-analyses. All of these capabilities have been checked out against available test data and other computer codes.

Calculation of nonisothermal cyclic material nonlinearities in the form of plasticity, creep, failure and fatigue are included in a manner conducive to the use of composite materials. Different material models can be assigned to different materials within the same element, allowing the material response in different layers to be calculated with appropriate isotropic or orthotropic material models. Access is provided to the nonlinear material iteration schemes so that user supplied material models can be included.

There is a requirement that the element shape follows the layup of the structure so that the layers cut through opposite faces of the element at the same relative locations and not diagonally across the element. This requirement points to the use of a mesh generator, which is a part of the CSTEM structural module. The CSTEM mesh generator is capable of producing various solids of revolution from a minimum of input parameters. The generator can produce flat surfaces, cylinders, cones, and general double curved surfaces of up to 360 degrees rotation. These different surface types can be generated together. The generator will check for coincident nodes and keep only one of any nodes which have the same coordinates. When generating more than one surface, care must be taken that the surfaces are generated so that the connecting nodes will have the same coordinates.

Where more complex models are required, the CSTEM analysis preference in the PATRAN code allows the user to build the analysis model, define materials and material model properties, apply boundary conditions, select CSTEM analysis options, and generate a complete CSTEM input deck directly from within PATRAN. Analysis results may then be displayed within PATRAN as well.

Another capability that can be used together with the multiple layer capability is a composites analyzer, which is adapted from the ICAN computer program. This capability must also be used together with the INHYD generation of material properties. The composites analyzer takes the stress/strain results from the structural module and integrates them through the thickness of the structure at some user specified location. This results in a loading which can be used by the composites analyzer to do a micro-analysis of the composite at that particular location. Additionally, enhanced

interlaminar stress calculations can be requested to take into account the strain discontinuities present between composite material layers.

The heat transfer module has the capability to perform four different analysis types. In increasing order of complexity these are linear steady state, nonlinear steady state, linear transient, and nonlinear transient. The types of loadings that can be used in the heat transfer analyses are nodal and surface heat fluxes, convection, radiation, and internal heat generation. The material thermal conductivity and specific heat are required as material properties. Table 1. lists the parameters involved in the various heat transfer analysis types.

TABLE 1. THERMAL ANALYZER PARAMETERS

Thermal Parameters and Boundary Conditions	Steady State		Transient	
	Linear	Nonlinear	Linear	Nonlinear
Temperature	T	T	T(t)	T(t)
Time	—	—	t	t
Thermal Conductivity	$k_{ij}$	$k_{ij}(T)$	$k_{ij}(t)$	$k_{ij}(T,t)$
Convection Coefficient	h	$h(T)$	$h(t)$	$h(T,t)$
Internal Heat Generation	$Q_i$	$Q_i$	$Q_i(t)$	$Q_i(t)$
Surface Heat Flux	$Q_s$	$Q_s$	$Q_s(t)$	$Q_s(t)$
Convection Boundary	$Q_c$	$Q_c$	$Q_c(t)$	$Q_c(t)$
Specified Nodal Temperatures	$T_s$	$T_s$	$T_s(t)$	$T_s(t)$
Heat Capacity	—	$C_p(T)$	—	$C_p(T,t)$
Radiation Emissivity	—	$\epsilon (T)$	—	$\epsilon (T, t)$
Viewing Factor	—	f	—	f(t)

In the nonlinear analyses, material properties are entered at several temperature points and interpolated to the calculated temperatures. The solution is iterated upon until convergence is achieved.

In the transient analyses, time steps are specified and boundary conditions are entered at each of these time steps. The solution is obtained and printed out at each time step point by stepping along a series of evenly spaced user specified time subincrements.

When doing the heat transfer analysis as part of a coupled solution, the calculated nodal temperatures are passed to the structural module so that the structural material properties and thermal strains are calculated using these temperatures. A structural solution can be obtained at specific time step points as requested by the user input in the case of a transient heat transfer analysis.

The layering capability of the structural module is also used in heat transfer. This and the ability to specify orthotropic material thermal conductivities provides the capability to perform accurate heat transfer analyses of composite materials. The INHYD micromechanics program for generating laminate material properties from the material constituents is also available from the heat transfer module to generate thermal properties.

The electromagnetic absorption module has three options for calculation of absorption or attenuation of electromagnetic energy. All three methods use a data bank of absorption material properties. The first method uses the computer program WAVES as a subroutine in CSTEM. This program calculates the reflection and transmission of electromagnetic waves given a layup sequence of materials and their electromagnetic properties. Using this layup the WAVES program develops complex impedances to calculate reflection and transmission coefficients for the cross section. The electromagnetic properties needed are complex values of the permittivity and permeability, which are obtained from the data bank.

The second method calculates reflection, refraction, and attenuation of electromagnetic waves by using Snell's Law, the Fresnel Formulas, and the attenuation constant as derived from the vector wave equations. This method requires the material properties of permittivity, permeability, and conductivity to be available as a function of temperature and frequency on a data bank similar to the first method.

The third method uses a data bank that is different from the first two methods in that it contains absorption properties for the material at not only discrete values of temperature and frequency, but also polarization angle. The absorption of electromagnetic energy of a specific frequency and polarization by a given material at a specific temperature is calculated by linearly interpolating from the discrete data bank values.

The orientation of an electromagnetic wave is specified similar to a skew material so that a coordinate system is associated with the wave propagation. This wave coordinate system is defined such that the direction of propagation is along the positive Z axis and polarization is measured from the positive X axis. The orientation of the wave coordinate system with the global coordinate system is specified using skew transformations.

The element face upon which the electromagnetic wave is impinging is specified by the input. The path taken through the structure thickness is determined by the program assuming that the wave always exits through the opposite element face that it entered. Absorption calculations are made for each material encountered and are carried out using midsurface centroid values of temperature and

orientation. The impingement angle is calculated as a dot product of the wave coordinate system Z axis and the midsurface centroid normal. The polarization angle is calculated as the dot product of the projection on the layer midsurface of the wave polarization and the material orientation.

Absorption calculations are done for one given frequency, orientation, and wavepath at a time. If it is necessary to calculate results for several frequencies, orientations or wavepaths, a separate calculation must be done for each combination.

The approach to calculate acoustic characteristics due to structural vibration in CSTEM determines the radiation efficiencies of a structure for each vibration mode as a function of frequency. An eigen-analysis produces the fundamental modes and mode shapes. Once the radiation efficiencies for each mode are calculated, the total sound power is obtained by a modal summation of the contribution from each mode.

CSTEM tailoring capability has been built on the STAEBL computer program obtained from NASA Lewis. This program consists of two major modules: CONMIN, which performs the actual tailoring, and ANALIZ which supplies the parameters to be tailored. The CONMIN module was abstracted from STAEBL and coupled with the CSTEM structural, thermal, electromagnetic, and acoustic analysis modules.

Figure 1. contains a flowchart of the major analysis modules of CSTEM. These modules are used as a stand alone analysis package with entry through the main executive routine, or as the analysis portion of the tailoring process in which case the entry to these modules is at the load case level. Figure 2. is the flowchart of the tailoring process itself.

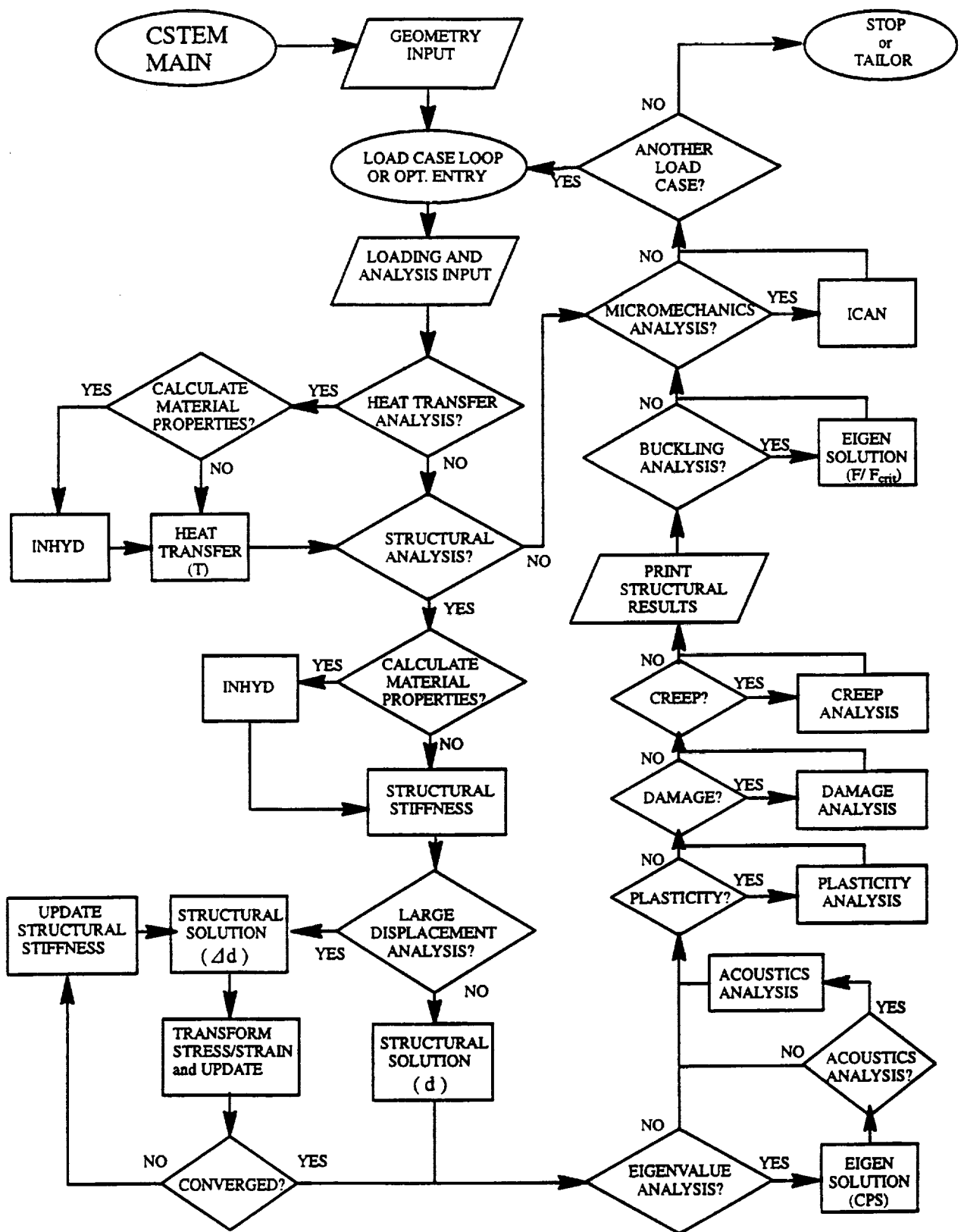


FIGURE 1. CSTEM FLOWCHART

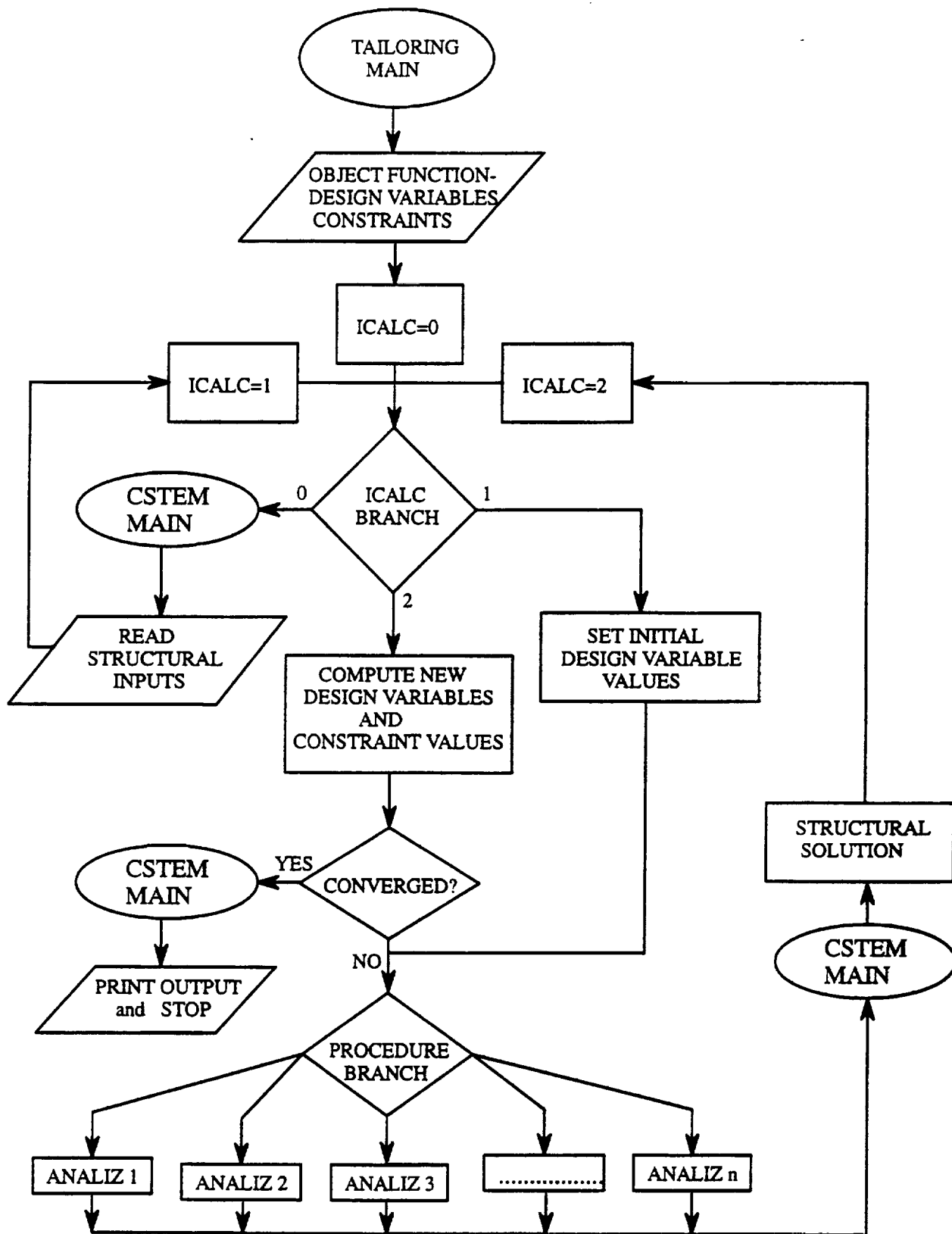


FIGURE 2. TAILORING PROCESS FLOWCHART





## **SECTION 2. MODELING CONSIDERATIONS**

### **2.1. COORDINATE SYSTEMS**

#### **2.1.1. Global Coordinate System (XYZ)**

The CSTEM code refers to several coordinate systems, all of which are right hand cartesian type of systems. The common point of reference for all systems is the global coordinate system (XYZ). This is the coordinate system in which the model geometry is defined. In other words, the nodes of a finite element mesh are points located by their global X, Y, and Z coordinates.

#### **2.1.2. Skew Coordinate Systems (xyz)**

Skew coordinate systems (xyz) can be set up for specified nodes and elements to simplify the specification of applied nodal boundary conditions and element orthotropic material properties. The skew coordinate systems are referenced throughout the code by a transformation number which is assigned when the skew system is created. There is no limit to the number of skew systems that can be specified, and they can be overwritten.

These skew systems are initially aligned with the global system and can be specified in one of four ways. Three of these four ways are by specific user input, while the fourth is a result of using the internal mesh generation capability. The first three ways are by reference to nodes (either structural nodes or reference nodes), direction cosines, or Euler angles. The fourth way makes use of the fact that a local coordinate system is generated for each node when generating the finite element mesh internally. These generated coordinate systems are related to the surfaces of revolution that can be created by internal meshing and can be saved for later use.

#### **2.1.3. Structural Coordinate System (rst)**

The structural coordinate system or element local coordinate system (rst) defines the element isoparametric space. The orientation of this coordinate system is determined by the element connectivity. This coordinate system is an orthogonal system in the isoparametric space of

the element. In global space, the structural system may not be orthogonal and the scale of each the individual axes may not be the same. This depends on the global geometry of the element. Figure 3. shows the structural coordinate system in global space where the node numbers are the location of the nodes in the element connectivity. The relationship between the global and structural systems is the Jacobian, which can vary throughout the element.

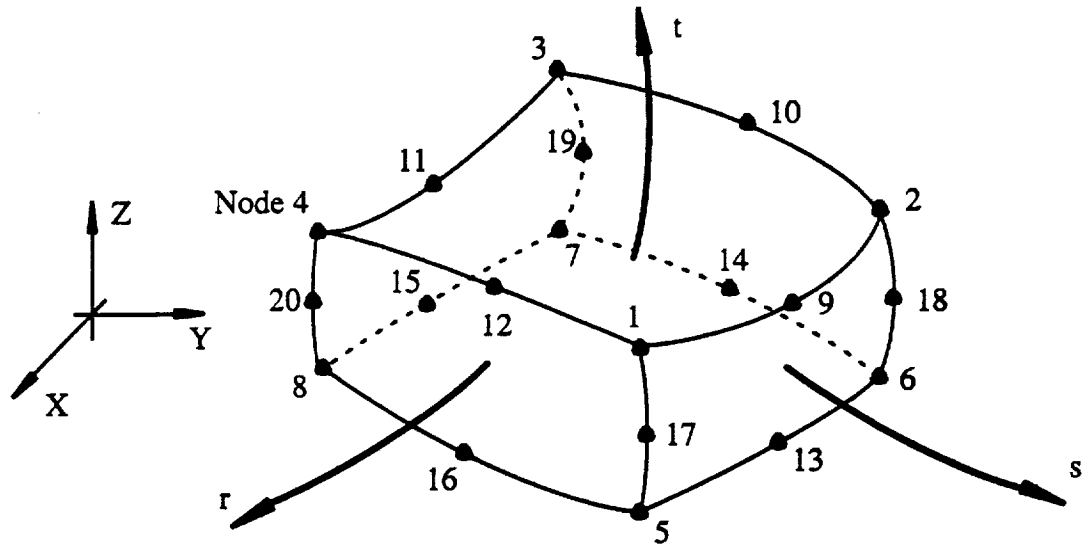


FIGURE 3. STRUCTURAL COORDINATE SYSTEM ORIENTATION FOR 3D ELEMENTS

The structural coordinate system can be used as a basis in orienting the material properties for the element. This is done by eliminating the scale transformation between the global and structural systems from the Jacobian so that only the angular transformation remains. Some modification to this angular transformation may have to be made to get an orthogonal transformation between the global to structural coordinate systems.

#### 2.1.4. Material Coordinate System (123)

The material coordinate system is relevant only for materials which are not isotropic since isotropic materials have the same properties regardless of orientation. For orthotropic materials, the properties are symmetric about three orthogonal planes, which defines a material coordinate

system. The material can be described by specifying the properties along each of the three axes of this system.

When using a composite material, the material principle direction is aligned with the material coordinate system 1 axis, the in plane transverse direction is aligned with the material 2 axis, and the out of plane transverse direction (through the thickness direction) is aligned with the material 3 axis. This is shown in Figure 4.

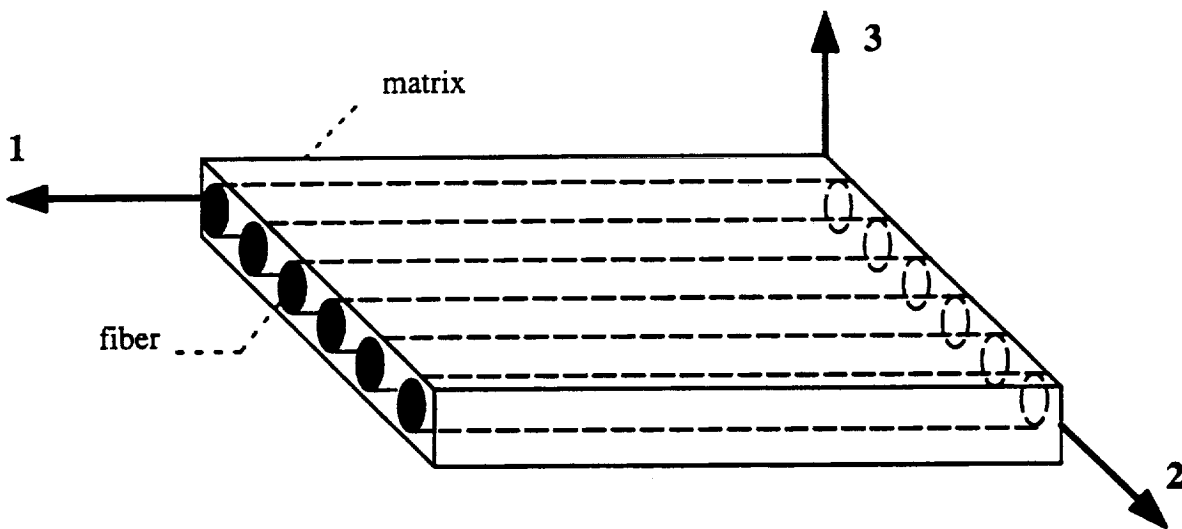


FIGURE 4. MATERIAL COORDINATE SYSTEM FOR COMPOSITE LAMINATE

The material properties are entered in the material coordinate system. The stiffness of the finite element model is calculated in the global coordinate system so the transformation between the material and global systems must be defined. This can be done directly by referencing a skew coordinate system transformation number or indirectly by specifying the transformation between the material coordinate system and structural coordinate system.

#### 2.1.5. Micromechanics Coordinate System (xyz)

The composite micromechanics analysis module (ICAN) has three coordinate systems associated with it. These are the composite structural coordinate system, the composite material coordinate system, and the ply material coordinate system as described in the ICAN user manual.

As used in CSTEM, the composite material coordinate system and the ply material coordinate system are the same and correspond to the material coordinate system described in 2.1.4. The composite structural coordinate system is basically the structural coordinate system of 2.1.3, with the restriction that the structural coordinate system z axis is aligned with the material 3 axis. This may be a permutation of the rst structural coordinate axis system of CSTEM since the t axis is not necessarily the axis that is aligned with the material 3 direction.

#### 2.1.6. Mesh Generation Coordinate System ( $x''y''z''$ )

The internal mesh generator in CSTEM is capable of generating various doubly curved surfaces of revolution. A local coordinate system that is initially aligned with the global system is rotated through 2 Euler angles so that the local  $x''$  axis passes through two specified points on the inner and outer surface of the mesh to be generated. The 2 Euler angles are rotations first about the global Z axis and then about the resulting  $y'$  axis. These angles are calculated from the input inner and outer surface points. The mesh is generated by rotating about the resulting local  $y''$  and  $z''$  axes. The local coordinate system used to generate a specific node can be saved for later use to define a skew coordinate system.

#### 2.1.7. Wave Coordinate System ( $xyz$ )

This coordinate system is used to define the orientation of electromagnetic waves in the electromagnetic analysis module of the CSTEM code. This is an orthogonal system defined such that the direction of propagation of the wave is along the z axis of the wave coordinate system. The polarization of the wave is referenced to the wave coordinate system x axis. Figure 5. shows the wave coordinate system. Orientation of the wave coordinate system with the global system is defined by specifying a skew coordinate system transformation number.

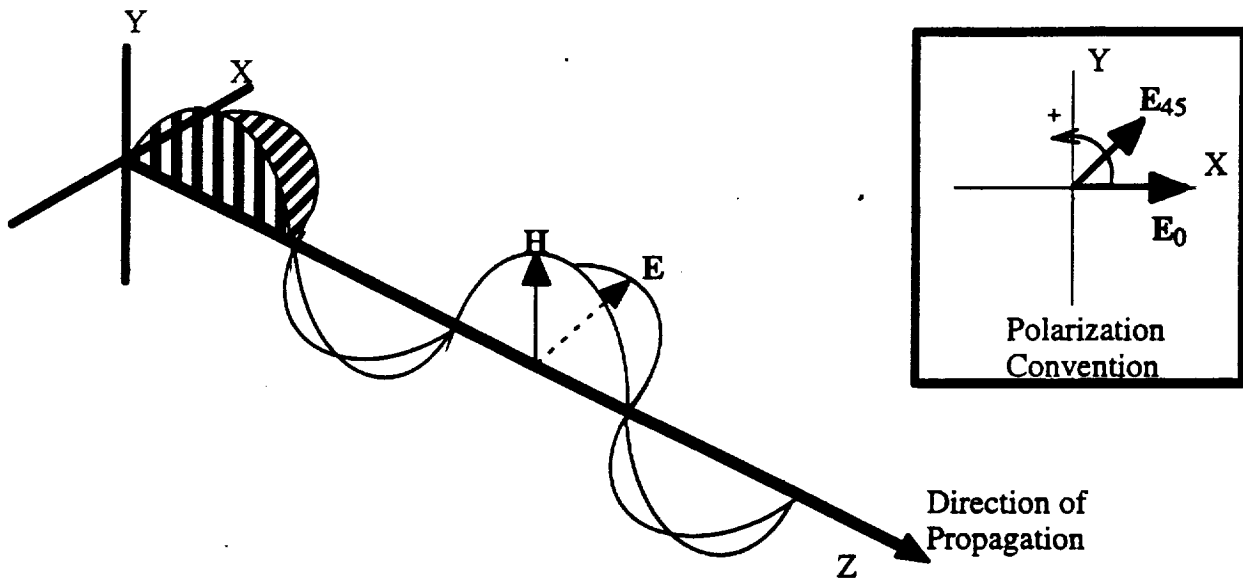


FIGURE 5. WAVE COORDINATE SYSTEM FOR 0 DEG. POLARIZED EM WAVE

## 2.2. SIGN CONVENTIONS

### 2.2.1. Angles

The coordinate systems used are right hand systems so the sign convention for angles is based on a right hand rule. The convention used is that positive angles are counterclockwise as viewed from the positive end of the axis of rotation. In right hand terms, if the thumb on the right hand is the axis of rotation, positive angles are in the direction of the curled fingers on the right hand when the thumb is placed on the end of the nose.

### 2.2.2. Forces/Displacements

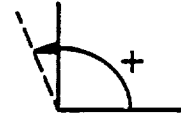
These quantities are signed according to the direction they act relative to the coordinate system they are referred to (usually global).

### 2.2.3. Pressures

Positive pressure causes compression on the face it is acting upon. In other words, positive pressure acts in the opposite direction of the face outward normal.

#### 2.2.4. Stress/Strain

Positive normal stress and strain is tensile, negative is compressive. Shear stress is positive when the right angle formed by the two axes defining the plane in which the shear stress acts is increased under the influence of the shear. (See schematic)



### 2.3. TRANSFORMATIONS

#### 2.3.1. Coordinate System Transformations

These type of transformations are used to change between different coordinate systems. Since the coordinate systems are all orthogonal this is basically a rotational transformation. Reference the books by Robert Cook on finite element analysis for further descriptions on details of transformations of this type.

##### 2.3.1.1 Coordinate Transformation

Coordinate transformations are done using a 3x3 orthogonal transformation matrix. The global XYZ coordinate vector is transformed to a local xyz coordinate vector by the equation

$$\{x\} = [T] \{X\} \quad (1)$$

Since the transformation matrix is orthogonal its inverse is simply the transpose so getting the local coordinates from the global is

$$\{X\} = [T]^T \{x\} \quad (2)$$

The transformation that is stored by CSTEM is that which starts with the global system and rotates it so that it ends up at the local system. The transformation matrix is a set of cosines of the angles between the local axes and the global axes as shown in Figure 6. Multiple global to local transformations are easily done by premultiplying the previous transformation matrix by the subsequent transformation matrix.

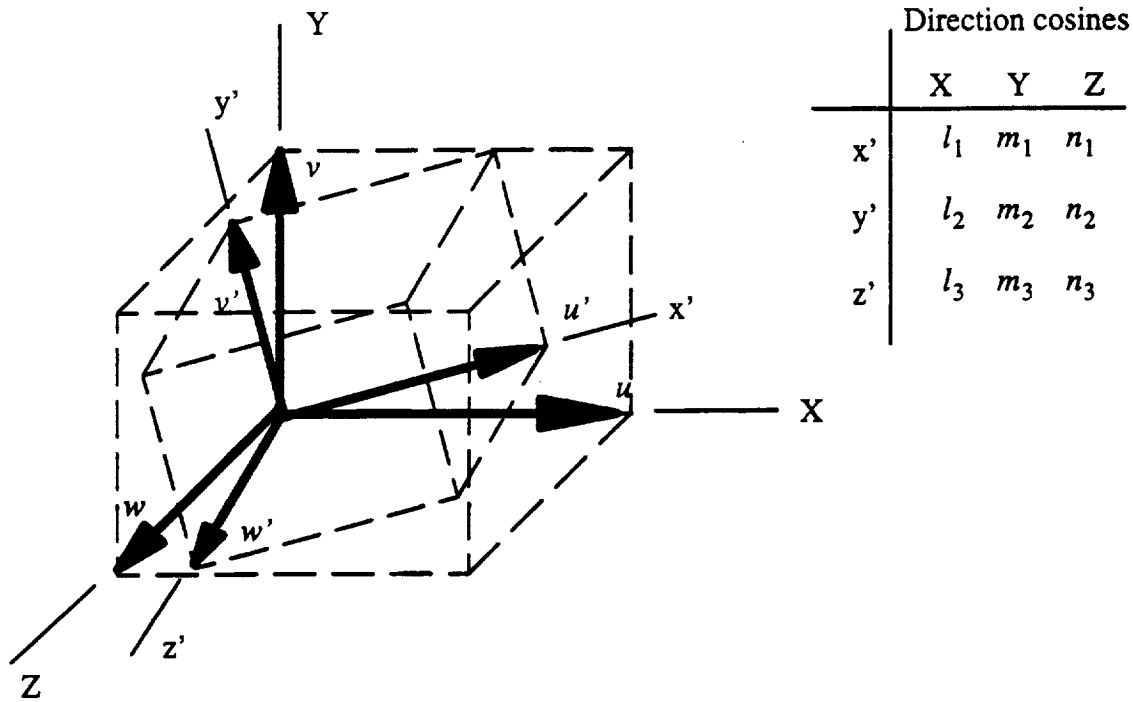


FIGURE 6. COORDINATE TRANSFORMATION MATRIX  
EXPRESSED AS DIRECTION COSINES

#### 2.3.1.2 Stress/Strain Coordinate System Transformations

The transformation matrix for stress is different than the transformation matrix for strain, although both are 6x6 matrices derived from the coordinate transformation matrix. They are also related in that the inverse of one is the transpose of the other. The equations transforming global strains and stresses to local are

$$\{\epsilon\}_L = [T_\epsilon] \{\epsilon\}_G \quad (3)$$

$$\{\sigma\}_L = [T_\epsilon]^{-T} \{\sigma\}_G \quad (4)$$

and from local to global are

$$\{\epsilon\}_G = [T_\epsilon]^{-1} \{\epsilon\}_L \quad (5)$$

$$\{\sigma\}_G = [T_\epsilon]^T \{\sigma\}_L \quad (6)$$

where Figure 7. shows  $[T_\epsilon]$ .

### 2.3.1.3 Material Coordinate System Transformations

The material transformation matrix is the 6x6 strain transformation matrix shown in Figure 7. The material properties are input in terms of the material coordinate system and are needed in terms of the global coordinate system for use in generating stiffnesses. The equation describing this transformation is

$$[D]_G = [T_\epsilon]^T [D]_L [T_\epsilon] \quad (7)$$

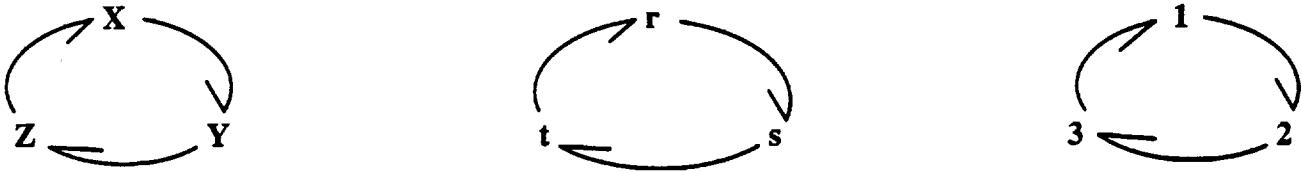
$$[T_\epsilon] = \left[ \begin{array}{ccc|ccc} l_1^2 & m_1^2 & n_1^2 & l_1 m_1 & m_1 n_1 & n_1 l_1 \\ l_2^2 & m_2^2 & n_2^2 & l_2 m_2 & m_2 n_2 & n_2 l_2 \\ l_3^2 & m_3^2 & n_3^2 & l_3 m_3 & m_3 n_3 & n_3 l_3 \\ \hline (2l_1 l_2) & (2m_1 m_2) & (2n_1 n_2) & (l_1 m_2 + l_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 l_2 + n_2 l_1) \\ (2l_2 l_3) & (2m_2 m_3) & (2n_2 n_3) & (l_2 m_3 + l_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 l_3 + n_3 l_2) \\ (2l_3 l_1) & (2m_3 m_1) & (2n_3 n_1) & (l_3 m_1 + l_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 l_1 + n_1 l_3) \end{array} \right]$$

FIGURE 7. STRAIN TRANSFORMATION MATRIX



#### 2.3.1.4 Coordinate System Permutations

This type of transformation is simply a labelling change on an existing coordinate system. This is used mainly in element layering when a structural coordinate system axis is designated as through the thickness, which corresponds to the material 3 axis. The structural coordinate axis corresponding to the material 1 direction is found by permutation. Permutations follow a 1,2,3 sequence and may best be visualized by the following diagram.



#### 2.3.2. Reference Configuration Transformations

These types of transformations are used in the updated Lagrange large displacement analysis. They transform between different geometry configurations. Reference the books of K.J. Bathe on finite element analysis for additional details on these types of transformations. A more detailed explanation may also be found in the section describing large displacement analysis. The configuration notation is that of Bathe where the leading subscript refers to the time to which the measure is referred and the leading superscript refers to the time when the measure is taken. This notation uses time only as a convenient scale between various configurations, not to imply a rate dependency.

##### 2.3.2.1 Deformation Gradient

Similar to the coordinate transformation matrix, all of the transformations to follow are derived from the deformation gradient. The deformation gradient relates the different configurations by taking partial derivatives of the coordinates of a configuration of time  $t$  with respect to coordinates of a reference configuration. Put in equation form using matrix notation, the

deformation gradient is

$${}^t_0[X] = ({}_0\nabla ({}^t_x)^T)^T \quad (8)$$

where

$${}_0\nabla = \begin{Bmatrix} \frac{\partial}{\partial {}^0x_1} \\ \frac{\partial}{\partial {}^0x_2} \\ \frac{\partial}{\partial {}^0x_3} \end{Bmatrix} \quad \text{and} \quad {}^t_x = \begin{Bmatrix} {}^tx_1 \\ {}^tx_2 \\ {}^tx_3 \end{Bmatrix} \quad (9)$$

and superscript T refers to the transpose. In tensor notation the deformation gradient is written

$${}^t_0X_{ij}$$

#### 2.3.2.2 Density Transformation

The determinant of the deformation gradient is the ratio of the densities of the different configurations.

$${}^0\rho = {}^t\rho | {}^t_0[X] | \quad (10)$$

#### 2.3.2.3 Strain Transformation

The Almansi strain is the current strain referenced to the current configuration and the Green strain is the current strain referenced to the original configuration. The deformation gradient transforms the Almansi to the Green strain according to the equation

$${}^t_0\epsilon_{mn} = {}^t_0X_{m,i} {}^t\epsilon_{ij} {}^t_0X_{n,j} \quad (11)$$

and the reverse relationship involves the inverse deformation gradient

$${}^t\epsilon_{mn} = {}^0X_{i,m} {}^t_0\epsilon_{ij} {}^0X_{j,n} \quad (12)$$

#### 2.3.2.4 Stress Transformation

The Cauchy stress is the current stress referenced to the current configuration and the 2nd Piola–Kirchoff stress is the current stress referenced to the original configuration. The deformation gradient transforms the 2nd Piola–Kirchoff stress to the Cauchy stress as

$${}^t\sigma_{mn} = \left(\frac{{}^t\rho}{0\rho}\right) {}^tX_{m,i} {}^tS_{ij} {}^tX_{n,j} \quad (13)$$

and the reverse relationship

$${}^tS_{mn} = \left(\frac{{}^t\rho}{0\rho}\right) {}^0X_{i,m} {}^t\sigma_{ij} {}^0X_{j,n} \quad (14)$$

#### 2.3.2.5 Material Transformation

The material matrix relating the Cauchy stress and Almansi strain can be transformed into the material matrix relating the 2nd Piola–Kirchoff stress to the Green strain with the equation

$${}^tC_{mnpq} = \left(\frac{{}^t\rho}{0\rho}\right) {}^tX_{m,i} {}^tX_{n,j} {}^0C_{ijrs} {}^tX_{p,r} {}^tX_{q,s} \quad (15)$$

### 2.4. ELEMENTS

#### 2.4.1. 3D Elements

The 3D elements used in CSTEM are isoparametric bricks with 8, 16, or 20 nodes. These element types are well documented and reference can be made to books by Cook, Bathe, and Zienkiewicz among others. Since the 8 and 16 node bricks are degenerated from the 20 node brick, the 20 node brick will be the basis of the following discussion.

The node connectivity used is that shown in Figure 3., where the node numbers shown are the connectivity positions, not the actual node number as defined by nodal input. The 20 node brick is shown, while the 8 and 16 node bricks would contain only the first 8 or 16 nodes respectively. The Jacobian is the transformation between the structural coordinate system

(isoparametric space) and the global coordinate system. Although the element can have a very irregular shape in the global system, it is a 2x2x2 cube in the structural system where

$$-1 \leq r \leq +1 \quad -1 \leq s \leq +1 \quad -1 \leq t \leq +1$$

The displacements anywhere within an element can be placed in terms of the displacements of the nodes. Let  $(u,v,w)$  be displacements at any point in the global  $(X,Y,Z)$  directions respectively, and  $(u_i,v_i,w_i)$ ,  $i=1$  to 20, be displacements of the element nodes in the global directions. The relationships between  $(u,v,w)$  and  $(u_i,v_i,w_i)$  are

$$u = \sum_{i=1}^{20} H_i u_i \quad v = \sum_{i=1}^{20} H_i v_i \quad w = \sum_{i=1}^{20} H_i w_i \quad (16)$$

where the  $H_i$  are the interpolation or shape functions.

The shape functions,  $H_i$ , are given as basic corner node functions which are modified by midside node functions. The 8 node brick has no midside node functions and the 16 node brick does not have all of them.

The basic corner node shape functions are

$$\begin{aligned} G1 &= (1+r)(1+s)(1+t)/8 & G5 &= (1+r)(1+s)(1-t)/8 \\ G2 &= (1-r)(1+s)(1+t)/8 & G6 &= (1-r)(1+s)(1-t)/8 \\ G3 &= (1-r)(1-s)(1+t)/8 & G7 &= (1-r)(1-s)(1-t)/8 \\ G4 &= (1+r)(1-s)(1+t)/8 & G8 &= (1+r)(1-s)(1-t)/8 \end{aligned} \quad (17)$$

The midside node shape functions are

$$\begin{aligned} H9 &= (1-r^2)(1+s)(1+t)/4 & H13 &= (1-r^2)(1+s)(1-t)/4 \\ H10 &= (1-r)(1-s^2)(1+t)/4 & H14 &= (1-r)(1-s^2)(1-t)/4 \\ H11 &= (1-r^2)(1-s)(1+t)/4 & H15 &= (1-r^2)(1-s)(1-t)/4 \\ H12 &= (1+r)(1-s^2)(1+t)/4 & H16 &= (1+r)(1-s^2)(1-t)/4 \\ H17 &= (1+r)(1+s)(1-t^2)/4 & H18 &= (1-r)(1+s)(1-t^2)/4 \\ H19 &= (1-r)(1-s)(1-t^2)/4 & H20 &= (1+r)(1-s)(1-t^2)/4 \end{aligned} \quad (18)$$

The modified corner node shape functions are then

$$\begin{aligned}
 H1 &= G1 - (H9 + H12 + H17)/2 \\
 H2 &= G2 - (H9 + H10 + H18)/2 \\
 H3 &= G3 - (H10 + H11 + H19)/2 \\
 H4 &= G4 - (H11 + H12 + H20)/2 \\
 H5 &= G5 - (H13 + H16 + H17)/2 \\
 H6 &= G6 - (H13 + H14 + H18)/2 \\
 H7 &= G7 - (H14 + H15 + H19)/2 \\
 H8 &= G8 - (H15 + H16 + H20)/2
 \end{aligned} \tag{19}$$

The first order definitions of strain in terms of displacements can be written as

$$\begin{aligned}
 \epsilon_{xx} &= \frac{\partial u}{\partial x} \\
 \epsilon_{yy} &= \frac{\partial v}{\partial y} \\
 \epsilon_{zz} &= \frac{\partial w}{\partial z} \\
 \epsilon_{xy} &= \frac{1}{2} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) = \frac{1}{2} \gamma_{xy} \\
 \epsilon_{yz} &= \frac{1}{2} \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) = \frac{1}{2} \gamma_{yz} \\
 \epsilon_{zx} &= \frac{1}{2} \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) = \frac{1}{2} \gamma_{zx}
 \end{aligned} \tag{20}$$

where the shear strains,  $\epsilon_{ij}$ , are the tensor shear strains and  $\gamma_{ij}$  are the engineering shear strains.

The engineering shear strains are used by CSTEM.

These equations can be written in matrix form as

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \end{Bmatrix} = \begin{bmatrix} \frac{\partial H_i}{\partial X} & 0 & 0 \\ 0 & \frac{\partial H_i}{\partial Y} & 0 \\ 0 & 0 & \frac{\partial H_i}{\partial Z} \\ \frac{\partial H_i}{\partial Y} & \frac{\partial H_i}{\partial X} & 0 \\ 0 & \frac{\partial H_i}{\partial Z} & \frac{\partial H_i}{\partial Y} \\ \frac{\partial H_i}{\partial Z} & 0 & \frac{\partial H_i}{\partial X} \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} \quad (21)$$

which, in a shorter notation is written as

$$\{\epsilon\} = [B]\{u\} \quad (22)$$

where  $\{\epsilon\}$  is the engineering strain vector,  $[B]$  is the strain-displacement matrix, and  $\{u\}$  is the vector of nodal displacements.

The terms of the strain-displacement matrix are partial derivatives of the shape functions with respect to the global coordinates; however, the shape functions were written in terms of the structural coordinates. To obtain the proper partials the chain rule of differentiation is used giving

$$\begin{aligned} \frac{\partial H_i}{\partial X} &= \frac{\partial H_i}{\partial r} \frac{\partial r}{\partial X} + \frac{\partial H_i}{\partial s} \frac{\partial s}{\partial X} + \frac{\partial H_i}{\partial t} \frac{\partial t}{\partial X} \\ \frac{\partial H_i}{\partial Y} &= \frac{\partial H_i}{\partial r} \frac{\partial r}{\partial Y} + \frac{\partial H_i}{\partial s} \frac{\partial s}{\partial Y} + \frac{\partial H_i}{\partial t} \frac{\partial t}{\partial Y} \\ \frac{\partial H_i}{\partial Z} &= \frac{\partial H_i}{\partial r} \frac{\partial r}{\partial Z} + \frac{\partial H_i}{\partial s} \frac{\partial s}{\partial Z} + \frac{\partial H_i}{\partial t} \frac{\partial t}{\partial Z} \end{aligned} \quad (23)$$

The Jacobian transforms the global coordinate system to the structural coordinate system.

$$\frac{\partial}{\partial \mathbf{r}} = [\mathbf{J}] \frac{\partial}{\partial \mathbf{X}} \quad (24)$$

where the Jacobian is

$$[\mathbf{J}] = \begin{bmatrix} \frac{\partial X}{\partial r} & \frac{\partial Y}{\partial r} & \frac{\partial Z}{\partial r} \\ \frac{\partial X}{\partial s} & \frac{\partial Y}{\partial s} & \frac{\partial Z}{\partial s} \\ \frac{\partial X}{\partial t} & \frac{\partial Y}{\partial t} & \frac{\partial Z}{\partial t} \end{bmatrix} \quad (25)$$

By comparing the partial derivative terms of the Jacobian and the equation above it can be seen that it is the inverse of the Jacobian that is necessary to calculate the strain-displacement matrix.

Integrations to calculate loads, stiffnesses, and volumes are performed in the structural coordinate system. These integrals take the form

$$F_{ij} = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} g_{ij} |J| dr ds dt \quad (26)$$

Specifically, the stiffness matrix is calculated as

$$[\mathbf{K}] = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} [\mathbf{B}]^T [\mathbf{D}] [\mathbf{B}] |J| dr ds dt \quad (27)$$

where  $[\mathbf{B}]$  is the strain displacement matrix and  $[\mathbf{D}]$  is the material constitutive matrix. The stiffness for each element is calculated separately and assembled into the global stiffness matrix.

Body forces are calculated using

$$F_{Bi} = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} [H_i]^T \{f_B\} |J| dr ds dt \quad (28)$$

where  $[H_i]$  are the shape functions and  $\{f_B\}$  is the vector of body force components. Again, the body forces are calculated for each element separately and assembled into a global force vector.

Initial strain effects can also be included as a force using

$$F_{0i} = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} [B]^T [D] \{\epsilon_0\} |J| dr ds dt \quad (29)$$

where  $\{\epsilon_0\}$  are the initial strains.

These integrals are evaluated by numerical integration of the form

$$F_{ij} = \sum_{a=1}^m \sum_{b=1}^n \sum_{c=1}^o g(r_a, s_b, t_c) |J| W_a W_b W_c \quad (30)$$

where  $m \times n \times o$  integration points are used and  $W_a, W_b, W_c$  are the weighting factors for locations  $r_a, s_b, t_c$ . CSTEM uses a Gauss–Legendre quadrature which employs an uneven distribution of integration points and weights to obtain an optimized solution to the integral. Table 2. gives the Gauss integration points and weights for integration orders up to 3. For a 20 node brick, which uses parabolic shape functions, an integration order of 3 gives an exact solution. An 8 node brick with its linear shape functions is exactly integrated by an order 2 Gauss quadrature.

Since the displacements are restricted to the shapes as given by the element shape functions, the stiffness generated by the finite element method is generally too stiff. Often a reduced integration order of 2 with 20 node bricks is used to counteract this tendency since reduced integration using Gaussian quadrature generally results in a more flexible stiffness. However, this may cause other problems, specifically that of zero energy modes in which erroneous



displacements which do not violate the stiffness of the structure may occur. Refer to books by Robert D. Cook, who gives a good explanation of this phenomena.

<u>Integration Order</u>	<u>i</u>	<u><math>r_i</math></u>	<u><math>W_i</math></u>
1	I	0	2.0
2	I	-.577350	1.0
	II	+.577350	1.0
3	I	-.774597	.5555555
	II	0	.8888888
	III	+.774597	.5555555

TABLE 2. GAUSS-LEGENDRE QUADRATURE POINTS AND WEIGHTS

#### 2.4.2. Equivalent Homogeneous Elements

Consider the case of laminated composites in which the laminates can be thin and of completely different properties in adjacent regions. Using a standard quadrature essentially requires the use of one element to be used for every layer in order to capture the non-continuous material changes. This results in an extremely fine mesh which translates to a large expense due to the large set of equations to be solved. Another option is to use an average material property that includes the effect of the material change. This gives a reasonable set of equations, but the stress and strain results as obtained at the integration points are averaged values because the material is an average of the ply materials and orientations. An additional micromechanics analysis could be done to obtain the true ply stresses and strains in this case.

The use of homogeneous elements for the analysis of layered composites in which several layers are contained within a single element requires the generation of an equivalent bulk orthotropic material. The equivalent material constitutive matrix is needed for use in the calculation of the element stiffness as shown in equation (27). This equivalent material constitutive matrix is obtained as a weighted sum of the global material matrices for each layer through the

element cross section, where the weighting is based on the fraction of the element thickness occupied by the layer as shown in equation (31). The numerical integration of the element stiffness is then carried out using a Gaussian distribution in all three directions.

$$[D]_{\text{bulk}} = \sum_1^{\text{\#layers}} [D]_{\text{layer}} \left( \frac{\text{thick}_{\text{layer}}}{\text{thick}_{\text{elem}}} \right) \quad (31)$$

Although the effect of differing layer stiffnesses can be accounted for with the use of a bulk material matrix, the effect due to the location of the different stiffnesses through the thickness of the element cannot be accounted for with this method. The bulk material matrix is the same at different integration points through the thickness of an element, but may vary from integration point to integration point when moving parallel to the layer plane. CSTEM provides the capability to use homogeneous elements separately or combined in a model with layered elements described in the following section.

#### 2.4.3. Layered Elements

An attractive third option in the analysis of laminated composites is to use layers of Gauss integration points in the direction of the changing material properties. Use of layered elements allows direct calculation of ply stresses without increasing the model size or pushing limits on element aspect ratio. This section deals with the elements which use layers of integration points in a single element. These elements can be used separately or can be combined with homogeneous elements within a model.

The layers contain integration points at the layer midsurfaces, using the usual Gauss quadrature locations on the isoparametric midsurface plane. This amounts to adding a fourth summation in the numerical integration scheme, where the fourth sum is over the number of layers within the element. One of the original three summations is also constrained to an integration order of 1, since the integration points are at the midsurface of each layer. The weight associated with the fourth summation is the element volume occupied by the layer being integrated. This can be

expressed as

$$F_{ij} = \sum_{l=1}^{nl} \sum_{a=1}^m \sum_{b=1}^n \sum_{c=1}^o g(r_a, s_b, t_c, \delta_l) |J| W_a W_b W_c Vol_l \quad (32)$$

where  $nl$  is the number of layers in the element,  $\delta$  is a thickness coordinate,  $Vol_l$  is the volume of the element occupied by the layer, and one of  $m$ ,  $n$ , or  $o$  has the value of 1 depending on which structural coordinate system axis is the through the thickness axis.

The displacement distribution through the thickness of the element is controlled by the shape function of the element. Layers of different thicknesses can be used, as well as different numbers of layers per element. However, consideration must be given to the capability to accurately integrate the differing stiffness properties of the layers. In general, layers of relatively equal thickness will provide a more accurate integration than a mixture of relatively thick and thin layers within an element. Figure 8. indicates the difference in stiffness integration as measured by the fundamental vibration frequency for a model with equal thickness layers relative to the usual 3rd order Gaussian distribution of integration points in 20 node brick elements. A 3rd order Gaussian distribution will give an exact integration for the regular 20 node brick elements of this model. It can be seen that even with 3 layers the results are within 6% of this exact integration. For an element which is detected to have only two layers, the CSTEM code will automatically split the two layers in half to create four layers.

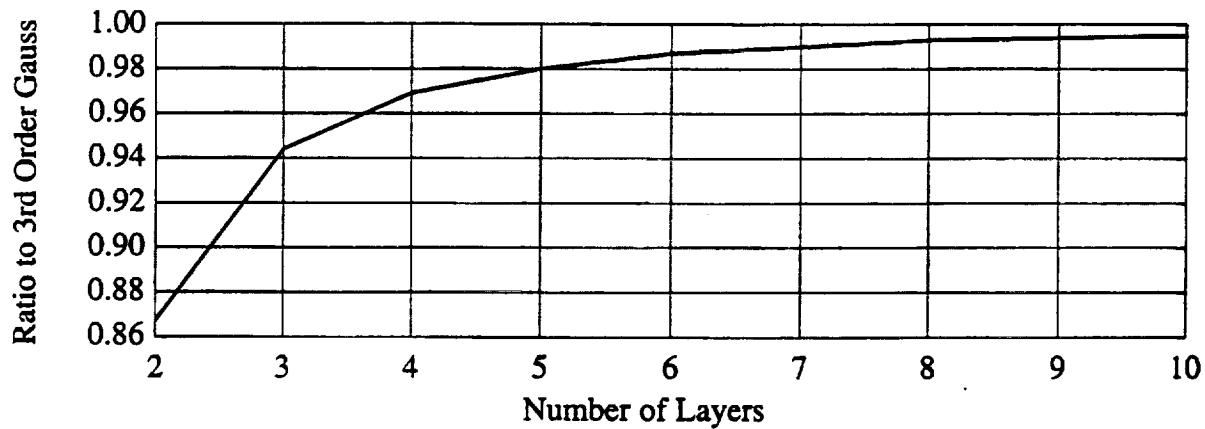


FIGURE 8. ACCURACY OF LAYERED INTEGRATION

The information necessary to define the layering of an element is the thickness axis, the layer material, the layer thickness, the number of layers, and the orientation of the layer. Multiple layers in CSTEM elements require that careful attention is paid to the orientation of the structural coordinate (or rst) system, determined by the element connectivity. In its isoparametric space an element is a  $2r$  by  $2s$  by  $2t$  cube with the rst system origin at the centroid. In other words the element sides (or faces) are planes perpendicular to one of the rst system axes and parallel to the plane defined by the other two axes. These element faces lie at the  $+1$  and  $-1$  locations of its perpendicular axis. The structural coordinate system definition for CSTEM is shown graphically in Figure 3., which is repeated below, and is also shown in Table 3.

LOCAL AXIS	FACE NUMBERS
$+t, -t$	1, 2
$+s, -s$	3, 4
$+r, -r$	5, 6

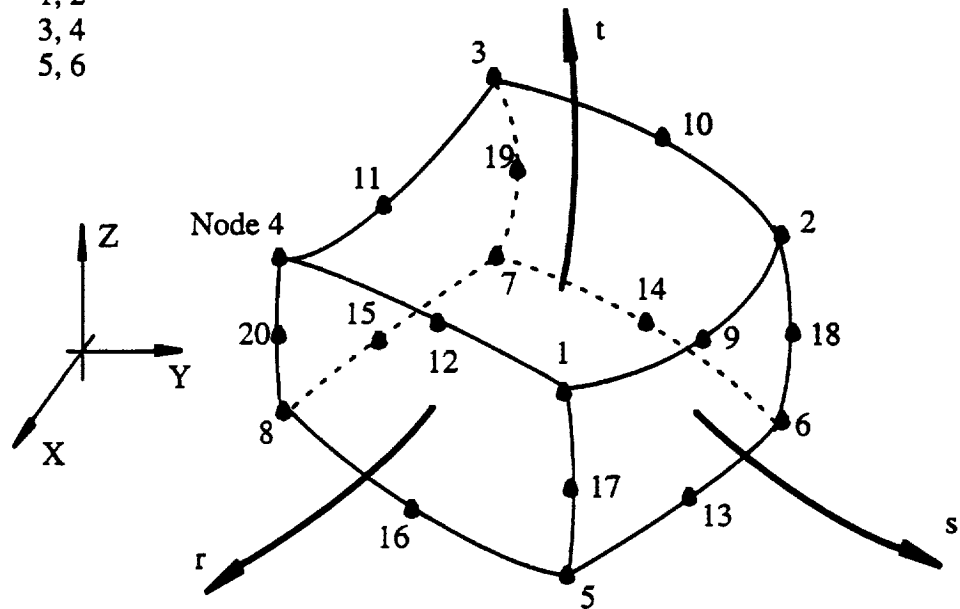


FIGURE 3. STRUCTURAL COORDINATE SYSTEM ORIENTATION FOR 3D ELEMENTS

TABLE 3. STRUCTURAL COORDINATE SYSTEM DEFINITION

AXIS	CORNER NODES			
+t	1	2	3	4
-t	5	6	7	8
+s	1	2	6	5
-s	3	4	8	7
+r	1	4	8	5
-r	2	3	7	6

Material layers within an element are set up so that the upper and lower surfaces of a layer are parallel to a pair of element faces in isoparametric space. This means that in isoparametric space the layer surfaces are parallel to a plane defined by two structural axes with the third axis being perpendicular to the layer surfaces. The perpendicular structural axis defines the thickness direction so that going through the element thickness means we are going perpendicular to the layers. This way of defining element layers means that when modeling a layered structure the mesh must follow the layer surfaces so that layers run between opposite element faces. Layers can not cut diagonally across an element. Figure 9. illustrates this concept.

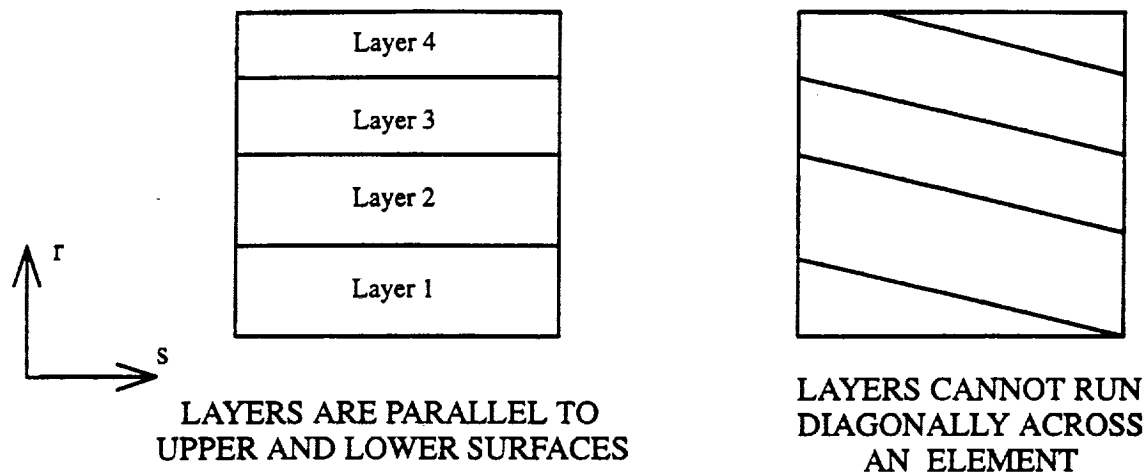


FIGURE 9. LAYERS IN AN ELEMENT

The layer material is defined by a material number and the properties associated with this material are stored by reference to this material number. Layer materials are treated as orthotropic materials, so isotropic materials can be used since they are a subset of orthotropics. Orthotropic materials have directionality so a material coordinate system is associated with each material. The material properties are specified with respect to this material coordinate system with the material X or 1 axis usually being the principle or stronger direction, although this is not a requirement.

Since the layers are of constant thickness in isoparametric space, the layer thicknesses are stored as fractions of the element thickness. Therefore, if the element varies in thickness in the global coordinate system then so do the layers. CSTEM is set up so that layer number 1 is at the negative end of the thickness axis, the 'lower' or 'bottom' surface of the element. The layers are then numbered consecutively progressing along the thickness axis in the positive direction.

There are two ways to define the orientation of the layer with respect to the global coordinate system. One is to define a transformation directly from the material coordinate system to the global coordinate system. In CSTEM a particular transformation has a number associated with it much like the materials. For global orientation of element layers the transformation number is all that is required and the transformation associated with that number is used to rotate the material properties directly to global.

Many structures are of complex shapes and curvatures which makes it difficult to specify the material orientation with respect to a global coordinate system. Since the elements in a mesh generally follow the shape of a structure (as do the layers) it may be much easier to specify the material orientation with respect to the element coordinate system. The material orientation with respect to the global system can then be calculated since the information relating the rst system to the global system is available from the element geometry definition. Using this method in CSTEM assumes that the material coordinate system is aligned so that the material Z or 3 axis is coincident with the element thickness axis.

For local orientation of element layers the angle of rotation about the material 3 axis is all that is required. The reference axis for this rotation angle depends on which of the rst axes is defined as the thickness axis. Table 4. shows the reference (0 degree) orientation of the material coordinate system to the element rst system for each thickness axis case.

	MATL AXIS	ELEMENT AXES		
reference axis (0 deg)	1 =	r	s	t
(90 deg)	2 =	s	t	r
thickness axis	3 =	t	r	s

TABLE 4. UNROTATED MATERIAL ORIENTATION WITH ELEMENT

The orientation of the structural coordinate system with respect to the global coordinate system is contained in the Jacobian matrix. The Jacobian contains both the angular relation of the element coordinate system to the global coordinate system as well as the stretching or shrinking of the element axes in relation to the global axes. For the material orientation only the angular relationship is required. Therefore, the Jacobian must be normalized row by row to get the transformation matrix from the element coordinate system to the global coordinate system. (If the

Jacobian were to be normalized column by column the transformation matrix from the global coordinate system to the element coordinate system would be obtained).

A possible drawback in using the Jacobian in this way is that the structural coordinate system may not be an orthogonal system when viewed in the global coordinate system. This would occur when element corners are not right angles. To avoid this problem an orthogonal system is set up based on the element coordinate system by taking successive cross products.

The default sequence of cross products is to cross the element axis corresponding to the material 3 direction into the element axis which corresponds to the material 1 axis. This forms the material 2 axis. The material 1 axis is then determined by crossing the 2 axis into the 3 axis. This sequence is such that the material 3 axis is indeed the element axis defined to be through the thickness, and the 3-1 plane defined by the element thickness axis and reference axis is preserved. Figure 10. illustrates the default sequence of cross products to form an orthogonal material coordinate system with the element  $r$  axis as the through thickness (material 3) axis.

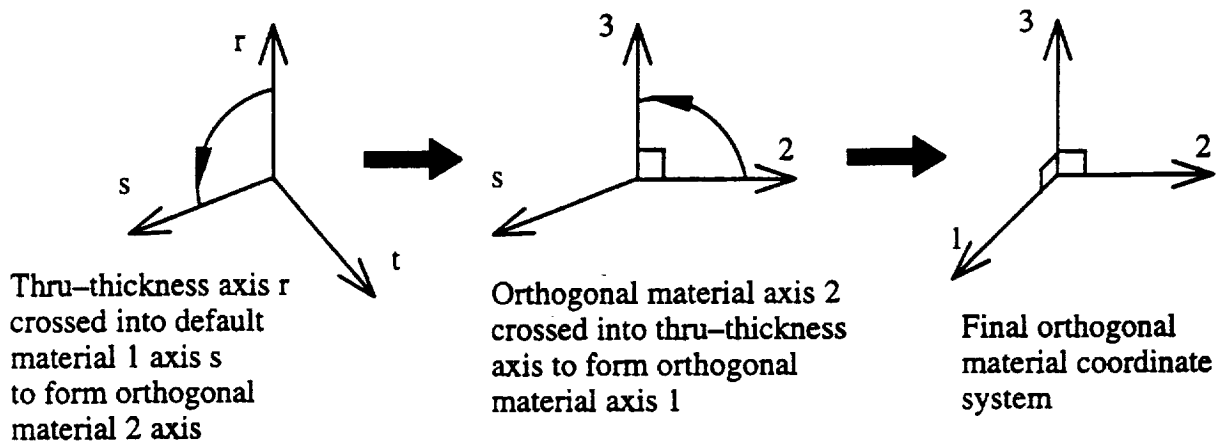


FIGURE 10. FORMATION OF ORTHOGONAL MATERIAL COORDINATE SYSTEM

Forming an orthogonal system eliminates some restrictions on the mesh, but still requires that the nodes defining the element thickness axis be lined up through the thickness. The default sequence of cross products can be changed if the through thickness axis is not orthogonal to the layer plane. This is indicated on input by a 2 digit code for the through thickness axis, the tens



digit being the through thickness axis and the ones digit being the axis which remains unchanged. If the material 1 axis is to remain unchanged, axis 1 is crossed into axis 2 to form axis 3, then axis 3 into axis 1 to form axis 2. If the material 2 axis is to remain unchanged, axis 1 is crossed into axis 2 to form axis 3, then axis 2 crossed into axis 3 to form axis 1. Both of these alternatives modify the material 3 axis from the corresponding element axis and preserve the material 1–2 plane.

Eliminating the requirement that elements have nodes lined up in the thickness direction is helpful, but the assignment of a material reference axis based on the element coordinate system places a requirement on the mesh that two faces must be parallel to the material reference plane. This may be too restrictive at times, as when meshing about a hole drilled through a composite plate, so the IGAX option is available to allow specification of a global axis that is to be parallel or perpendicular to the reference axis of the local orthogonal system. This way of specifying the material direction is a hybrid between a local and global method. If perpendicularity is requested, the angle between the specified global axis and the local reference axis is checked and if it is not 90 degrees the orthogonal system is rotated about the thickness axis so that it becomes 90 degrees. The IGAX option can also be used to request rotation about the thickness axis so that the material reference axis is parallel to a particular global axis. It can also be used to orient the material reference axis parallel or perpendicular to the tangential axis of a cylindrical coordinate system placed at the global origin.

The rotation of the material matrix from the material coordinate system to the global coordinate system is calculated as:

$$[D]_G = [T]^T [D]_L [T] \quad (33)$$

Therefore, to get the complete transformation matrix from the material coordinate system to the global coordinate system the normalized Jacobian (which rotates from element to global) is premultiplied by the planar transformation matrix which rotates the material coordinate system to the structural coordinate system.

#### 2.4.4. Additional Stiffnesses

Stiffnesses between nodal degrees of freedom can be input directly. These stiffnesses relate the translational and rotational degrees of freedoms of the nodes. This feature is useful for simulation of soft ground springs, allowing calculation of free-free vibration modes. Nodal masses and rotary inertias can be input for the same degrees of freedom related by the additional stiffnesses to simulate system effects.

### 2.5. CONSTRAINT EQUATIONS

Constraint equations define a relationship between degrees of freedom. The method used in CSTEM to impose the relationship specified by the constraint equation input is called the transformation method. The form of the constraint equation is shown in equation (34), where  $u_d$  is the dependant DOF,  $u_i$  is one of the  $i$  independant DOF,  $c_i$  is the coefficient for the  $i$ th independant DOF, and  $C$  is a constant.

$$u_d = C + \sum_i c_i u_i \quad (34)$$

## **SECTION 3. ANALYSIS TYPES**

### **3.1. STRUCTURAL ANALYSES**

#### **3.1. 1. Load Types**

Structural loadings for CSTEM can be input as nodal forces, nodal displacements and fixities, element pressures, centrifugal loads, acceleration loads, and nodal temperatures, which result in thermal loads. Once applied, all of these structural loading boundary conditions remain active from load case to load case until explicitly changed. Only rotational speed (RPM) must be input in every load case.

Nodal forces and displacements are applied in each degree of freedom of the nodal skew coordinate system, which is the global coordinate system until changed explicitly. It should be noted that nodal displacements are nonzero displacements and are applied by modifying the stiffness and the force vector so that the prescribed displacement results. This is in contrast to nodal fixities which are zero displacements and are incorporated by eliminating the fixed nodal degree of freedom from the global set of equations. Thus, a nodal fixity once defined can not be changed in an analysis, while a nodal displacement can be changed throughout the analysis.

Element pressures are applied to element faces, which are defined by the connectivity. The face numbering convention is shown in Table 5. Positive pressure values cause compression, acting into the face on which they are applied. In updated large displacement analyses, the pressure can be calculated using either the deformed or original face geometry. The pressure can vary across the face surface by specifying different pressure values at the face nodes. The force due to the pressure distribution across the element face is calculated by numerical integration using a two dimensional 3X3 Gauss quadrature. The nodal pressure values are interpolated at the integration points using the element shape functions.

Centrifugal loads are body forces calculated about one of the 3 global axes. The loads are calculated for a reference rotational speed of 1000 RPM and then scaled to the actual speed during force assembly. The mass density of the element is calculated during stiffness integration using the

	RST AXIS	N1	N2	N3	N4	N5	N6	N7	N8
FACE 1	+t	1	2	3	4	9	10	11	12
FACE 2	-t	5	6	7	8	13	14	15	16
FACE 3	+s	1	2	6	5	9	18	13	17
FACE 4	-s	3	4	8	7	11	20	15	19
FACE 5	+r	1	4	8	5	12	20	16	17
FACE 6	-r	2	3	7	6	10	19	14	18

TABLE 5. ELEMENT FACE NUMBERING CONVENTION

material density at each integration point of the element. This element density multiplied by the square of the reference rotational speed in radians per second is distributed as loads to the nodes using the shape functions and the fraction of the element volume associated with each integration point. Acceleration loads are handled similarly with the input acceleration load being used in place of the density times speed squared term.

To calculate thermal loads, the nodal temperatures are interpolated to the integration points using the element shape functions. The thermal strain at the integration point oriented in the material coordinate system is calculated using the material thermal expansion coefficients at this temperature. This thermal strain is transformed to the global coordinate system and integrated into an elemental thermal force using the equation for an initial strain force.

### 3.1. 2. Material Properties

Material properties can be specified in one of two ways in CSTEM. The properties can be input directly or they can be calculated from constituent properties using micromechanics. One or the other of these methods can be used, they can not be used together in the same analysis.

#### 3.1. 2. 1. Direct Input of Material Properties

When specifying material properties directly, the number of different material types that can be used together in CSTEM is unlimited since the material properties are stored on a random file and accessed by the material number associated with the properties. This material number is assigned to the elements and/or layers of that material. Isotropic and orthotropic materials can be used in CSTEM, at the same time if desired. The material properties are input at up to 10 discrete temperature levels in increasing order to describe the variation of these properties with temperature. The properties are linearly interpolated to the integration point temperature from the two bracketing material temperatures. If only one material temperature is input then those properties are used always. If the interpolation temperature is greater than the highest material temperature the properties are linearly extrapolated from those at the two highest material temperatures, and if the interpolation temperature is less than the lowest material temperature the properties are linearly extrapolated from those at the two lowest material temperatures.

#### 3.1. 2. 2. Calculation of Material Properties

For composite materials, ply material properties can be calculated from the constituent material properties for up to 10 different constituent combinations at a time. The constituent properties are contained in a data bank which must be available to the program. Reference can be made to the user manual for the NASA computer program ICAN (NASA Technical Paper 2515) for details on the micromechanics analysis performed in calculating these properties.

In conjunction with the calculation of ply properties using micromechanics, the capability also exists to group plies and create material properties which are actually a particular combination of ply orientations. This feature can be used as a way to simulate 2D composite weave properties.

#### 3.1. 3. Small Displacement Analysis

The basic type of structural analysis in CSTEM is the small displacement analysis. This assumes that all displacements are small relative to the model coordinates so that the original geometry can always be used to calculate stiffnesses and loads. The total loads are applied in each

load case to the system stiffness calculated for the temperature distribution of that load case. The results are total displacements. The equation used is

$$[K] \{u\} = \{F\} \quad (35)$$

which can be solved for the displacements giving

$$\{u\} = [K]^{-1} \{F\} \quad (36)$$

### 3.1. 3. 1. Stress Recovery

Stress recovery for small displacement analyses is performed element by element for each integration point contained within the element. Strains are calculated from the displacements of the nodes of an element as shown in equation (21). Stresses are calculated using

$$\{\sigma\} = [D]\{\epsilon\} \quad (37)$$

An alternative method for calculation of interlaminar shear stresses which uses stress equilibrium equations and in plane stresses calculated as shown in equation (37) is also available. See section 3.1. 10. for more details.

#### 3.1. 3. 1.1. Failure Criteria

As part of the stress recovery procedure, various failure criteria can be calculated. Currently, the available failure criteria are the Tsai–Wu quadratic criteria, the Max Stress criteria, the Max Strain criteria, and a modified Tsai–Hill failure theory. The particular failure criteria to be used is associated with a material type by input. The stress/strain orientation to which the failure strengths are applied is determined by the orientation requested by variable IPOUT. If stress/strain output is requested in global, the strengths are applied to global stress/strain values and if stress/strain output is requested in material, the strengths are applied to material stress/strain values. If principle stress/strain output is requested, strengths are applied to global stress/strain values.

The Max Stress and Max Strain criteria are applied by comparing the failure strength at the current integration point temperature with the calculated stress and strain. This is done on a

component by component basis, with the maximum component failure criteria value being the one reported for the integration point. The Tsai–Wu quadratic failure criteria is calculated using

$$F_{11}(\sigma_{11})^2 + F_{22}(\sigma_{22})^2 + F_{33}(\sigma_{33})^2 + F_{44}(\sigma_{12})^2 + F_{55}(\sigma_{23})^2 + F_{66}(\sigma_{31})^2 + F_1\sigma_{11} + F_2\sigma_{22} + F_3\sigma_{33} + 2F_{12}\sigma_{11}\sigma_{22} + 2F_{23}\sigma_{22}\sigma_{33} + 2F_{13}\sigma_{11}\sigma_{33} = 1 \quad (38)$$

where  $F_{ii} = \frac{1}{S_{ii}S'_{ii}} \quad ; \quad F_i = \frac{1}{S_{ii}} - \frac{1}{S'_{ii}} \quad ; \quad F_{ij} = \frac{1}{(S_{ij})^2}$

and S is a tensile strength and S' is a compressive strength.

The Tsai–Hill failure theory involves only in plane components of stress and strength, and can be written as

$$\left(\frac{\sigma_{11}}{S_{11}}\right)^2 - \frac{\sigma_{11}\sigma_{22}}{S_{11}S_{22}} + \left(\frac{\sigma_{22}}{S_{22}}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 = 1 \quad (39)$$

where S is a tensile strength.

#### 3.1. 4. Large Displacement Analysis

The large displacement analysis method used in CSTEM is an incremental updated Lagrange method, in which the nodal coordinates are continuously updated to the current equilibrium position and all stress and strain measures are referred to this updated configuration. This method handles the effects of previous displacements automatically due to the fact that the nodal coordinates include the displacements.

The basic equations used are derived from consideration of equilibrium, stating that the externally applied forces must be balanced by the stresses generated internally. For this balance to be correctly achieved mathematically, care must be taken to use the proper stress and strain measures. For updated Lagrange these are the Cauchy stress and Almansi strain, as opposed to the 2nd Piola–Kirchoff stress and Green strain which use a previous configuration as their reference basis.

The Almansi strain is the strain referenced to the current configuration. It is defined as:

$${}^t\epsilon_{ij} = \frac{1}{2} ( {}^tu_{i,j} + {}^tu_{j,i} - {}^tu_{k,i} {}^tu_{k,j} ) \quad (40)$$

where tensor notation is used. In the previous equation, the left superscript refers to the time when the strain occurs and the left subscript refers to the configuration to which the strain is referenced. This time notation is used throughout this manual.

The Green strain is the strain referenced to the original configuration. It is defined as:

$${}_0\epsilon_{ij} = \frac{1}{2} ( {}_0u_{i,j} + {}_0u_{j,i} - {}_0u_{k,i} {}_0u_{k,j} ) \quad (41)$$

The relationship between the Almansi strain and the Cauchy stress is

$${}^t\sigma_{mn} = {}^tC_{mnpq} {}^t\epsilon_{pq} \quad (42)$$

and between the Green strain and 2nd Piola–Kirchoff stress

$${}_0S_{ij} = {}_0C_{ijrs} {}_0\epsilon_{rs} \quad (43)$$

These stress and strain measures with respect to the current and original configurations can be related to each other through the use of the deformation gradient. Extensive use is made of this since stress and strain increments calculated in previous loading increments must be transformed to the current configuration and summed together to obtain the current stress/strain state. The deformation gradient is defined in matrix notation as

$${}_0[X] = ({}_0\nabla {}^t\{x\}^T)^T \quad (44)$$



where

$${}_0\nabla = \begin{Bmatrix} \frac{\partial}{\partial {}^0x_1} \\ \frac{\partial}{\partial {}^0x_2} \\ \frac{\partial}{\partial {}^0x_3} \end{Bmatrix} \quad \text{and} \quad {}^t\{x\} = \begin{Bmatrix} {}^tx_1 \\ {}^tx_2 \\ {}^tx_3 \end{Bmatrix} \quad (45)$$

and superscript T refers to the transpose.

Using tensor notation the relations between stress and strain measures can be written as

$${}^t\sigma_{mn} = \left(\frac{{}^t\rho}{{}_0\rho}\right) {}^tX_{m,i} {}^tS_{ij} {}^tX_{q,s} \quad (46)$$

$${}^t\epsilon_{mn} = {}^tX_{i,m} {}^t\epsilon_{ij} {}^tX_{j,n} \quad (47)$$

$${}^tC_{mnpq} = \left(\frac{{}^t\rho}{{}_0\rho}\right) {}^tX_{m,i} {}^tX_{n,j} {}^tC_{ijrs} {}^tX_{p,r} {}^tX_{q,s} \quad (48)$$

where  $\rho$  is the density and

$${}_0\rho = {}^t\rho |{}_0[X]| \quad (49)$$

The equilibrium equations to be solved in the incremental updated Lagrange large displacement method are

$$({}^t[K_L] + {}^t[K_{NL}]) \{\Delta u\}^{(i)} = {}^{t+dt}\{R\} - {}^{t+dt}_{{}^{t+dt}}\{F\}^{(i-1)} \quad (50)$$

where

$${}^t[K_L] = \int_{V} {}^t[B_L]^T {}^t[C] {}^t[B_L] d^tV \quad (51)$$

$${}^t[K_{NL}] = \int_{V} {}^t[B_{NL}]^T {}^t\{\sigma\} {}^t[B_{NL}] d^tV \quad (52)$$

$${}^{t+dt}_{t+dt}\{F\} = \int_{V} {}^t[B_L]^T {}^t\{\sigma\} d^tV \quad (53)$$

and  ${}^{t+dt}\{R\}$  are applied loads at time  $t+dt$ .

In this notation, the left superscript and subscript indicate the calculation time and reference configuration as before, and the right superscript in parenthesis indicates the iteration number.

The linear strain displacement matrix,  $[B_L]$ , is the same as for the small displacement case. The nonlinear strain displacement matrix,  $[B_{NL}]$ , involves the same terms as  $[B_L]$ , but in a different arrangement. Reference can be made to the book by K.J. Bathe, Finite Element Procedures In Engineering Analysis.

The total displacement, strains, and stresses are the sum of the increments of each of these quantities. However, the stress and strain increments must all be with reference to the same configuration before summing.

### 3.1. 5. Finite Element Implementation of Large Displacement Analysis

The large displacement analysis method used in CSTEM is an incremental approach rather than the total approach used for small displacement analysis. In addition to the force calculation and assembly, the stiffness calculation is different from that used for a small displacement analysis as well as the stress recovery procedure and the use of equilibrium iteration.

### 3.1. 5. 1. Deformation Gradient

The deformation gradient, shown in equation (44), is used throughout a large displacement analysis. It is used to transform stresses, strains and the material matrix between various configurations.

The deformation gradient is formed for each integration point since it is used in integrating element stiffness and transformation of stresses and strains, which are calculated at the integration points. The first step in forming the deformation gradient is to calculate the Jacobian using the reference configuration coordinates of the element containing the particular integration point under consideration. If the reference configuration is time 0, the coordinates used in the Jacobian calculation would be the original undeformed coordinates. If the reference configuration is at some time  $t$ , the coordinates used would be the original coordinates plus the total displacements at time  $t$ . This Jacobian is then used to transform the derivatives of the shape functions, which are calculated with respect to the structural (rst) coordinate system, to the global coordinate system. These shape function derivatives with respect to the global coordinates at the current reference configuration are multiplied by the coordinates of the element at the desired calculation time, say  $t+dt$ . The resulting set of partial derivatives of the current coordinates with respect to the reference coordinates is the deformation gradient between the reference configuration and the current configuration.

### 3.1. 5. 2. Stiffness Calculations

As indicated by equations (50)–(52), the stiffness is composed of a linear and a nonlinear component. Since the large displacement analysis follows an updated Lagrange approach the updated coordinates are used in all stiffness calculations, the effect being included through the strain displacement matrix. In calculating the linear stiffness component, the material matrix must be updated from the original undeformed configuration to the current configuration using the deformation gradient, as shown in equation (48). The nonlinear stiffness component includes the integration point Cauchy stress, which involves a deformation gradient transformation as shown in equation (46).

### 3.1. 5. 3. Force Calculations and Assembly

Applied loads are generally calculated in the same way as in the small displacement case, (see equations (28) and (29)), except that updated coordinates are used. The internal force is calculated using equation (53), in which the stresses have been transformed to the configuration at the beginning of the load case. All loads are calculated and saved on an elemental level and later assembled into a global applied force and global internal force. Thermal effects are accounted for by saving the previous load case thermal load vector and assembling it into the internal force. The current load case thermal load vector is included as an applied load. The global incremental force is obtained by subtracting the internal force from the applied force as shown in equation (50).

### 3.1. 5. 4. Equilibrium Iterations

Equilibrium iterations can be done if desired so that the force imbalance due to the difference between the configuration at the beginning of the load case and the end of the load case can be eliminated to some tolerance. Equilibrium iteration convergence is based on the largest degree of freedom difference in the displacement increments from one iteration to the next.

When performing equilibrium iterations, the stiffness is recalculated at the beginning of each iteration with the previous displacement increment included in the updated configuration. The material matrix and stress included in the stiffness calculation are also transformed to the updated configuration. The reference configuration remains the configuration at the beginning of the load case.

The applied loads in the form of centrifugal forces, acceleration forces, and pressure forces are updated for each iteration as the configuration changes. The updated internal force is calculated at each iteration based on the updated stress state as well as the new configuration, so that the incremental load for the next iteration can be formed. The incremental load vector is modified to enforce prescribed displacements in each iteration. Thermal loads are applied in the initial load increment only and are not included in subsequent iterations.

### 3.1. 5. 5. Stress Recovery

Stress recovery is done differently for large displacement than in the small displacement case. A displacement increment is calculated for each applied load increment. The total strain increments are calculated at the individual integration points using equation (40), in which the higher order terms are included if requested. The elastic strain increment is found by subtracting the thermal strain increment, and the stress increment can then be calculated using equation (42). The total strain and stress state is obtained as a summation of the stress and strain increments calculated in each load increment.

When summing a stress and strain increment with any previous stress and strain state, the reference configuration of the increment and previous state must be the same. Since the increments are Almansi strains and Cauchy stresses, which are based on the current configuration, the previous stresses and strains are transformed to the current configuration using the deformation gradient as shown in equations (46) and (47). This transformation involves only the diagonal terms of the deformation gradient when a first order large displacement analysis is requested.

### 3.1. 6. Principal Stress Calculations

There are two options involving calculation of principal stresses and principal strains in CSTEM. Principal strains are calculated similarly using tensor shear strains. The first option is the usual 3D principal stress calculation, which solves for the eigenvalues of the 3D stress tensor. This calculation results in the solution of a cubic equation with stress invariants as coefficients of the equation. This will result in three normal stress components and no shear components. The stress or strain tensor is oriented in 3D space to obtain this condition.

$$\sigma^3 - I_\sigma \sigma^2 - \Pi_\sigma \sigma - \text{III}_\sigma = 0 \quad (54)$$

where

$$I_\sigma = \sigma_x + \sigma_y + \sigma_z$$

$$\Pi_\sigma = -(\sigma_x \sigma_y + \sigma_y \sigma_z + \sigma_z \sigma_x)$$

$$\text{III}_\sigma = \sigma_x \tau_{yz}^2 + \sigma_y \tau_{xz}^2 + \sigma_z \tau_{xy}^2 - (\sigma_x \sigma_y \sigma_z + 2\tau_{xy} \tau_{yz} \tau_{zx})$$

The other principal stress calculation is intended for use with layered materials, in which the principal stresses in the plane of the layer of the material are calculated while the out of plane

stress or strain components are unchanged, so this is a 2D principal stress calculation. In this case there will be two in-plane normal stress components and no in-plane shear component. The stress tensor is rotated about the material through thickness (3) axis to obtain this condition.

$$\sigma_{\max} = \left( \frac{\sigma_x + \sigma_y}{2} \right) + \tau_{\max} \quad \sigma_{\min} = \left( \frac{\sigma_x + \sigma_y}{2} \right) - \tau_{\max} \quad (55)$$

$$\text{where} \quad \tau_{\max} = \sqrt{\left( \frac{\sigma_x - \sigma_y}{2} \right)^2 + \tau_{xy}^2}$$

It is also possible to rotate the stress tensor about the material through thickness (3) axis so that only one interlaminar shear stress component exists and is therefore maximized, although the orientation of this shear component may lie at some angle to the fibers.. This calculation is separate and independent of the in-plane stress or strain values, but provides an alternate view of the interlaminar shear stress.

For a rotation about the material 3 axis:

$$\tau_{23}' = \tau_{23} \cos \alpha - \tau_{31} \sin \alpha \quad (56)$$

For  $\tau_{23}'$  to be zero:

$$\tau_{23} \cos \alpha = \tau_{31} \sin \alpha$$

$$\alpha = \tan^{-1} \frac{\tau_{23}}{\tau_{31}}$$

### 3.1. 7. Free Vibration Analysis

The free vibration analysis solves for the frequency of vibration and mode shapes for a specified number of modes up to a maximum number specified by the parameter MXROOT. Free vibration frequencies and mode shapes can be calculated using either the determinant search method or the subspace iteration method. Of the two methods, the subspace iteration method appears to be the more accurate and economical. This is because the determinant search method, using polynomial iteration as well as vector inverse iteration, is intended more for small banded systems of equations. The subspace iteration method uses the more economical vector inverse iteration, and performs the vector inverse iterations simultaneously for a number of the required

eigenvectors. This is especially efficient if the system is large enough to require out of core solution.

The generalized eigenproblem for free vibration analysis is

$$[K][\Phi] = \{\lambda\}[M][\Phi] \quad (57)$$

where  $[K]$  is the stiffness matrix,  $[M]$  is the mass matrix,  $[\Phi]$  is an array of eigenvectors, and  $\{\lambda\}$  is a vector of eigenvalues. Each eigenvector in  $[F]$  corresponds to a particular modeshape and the corresponding eigenvalue in  $\{\lambda\}$  is the free vibration frequency squared in radians per second.

Normally the stiffness matrix in equation (57) is the linear stiffness based on the original geometry. However, when a free vibration analysis is requested with large displacement this stiffness is based on the updated geometry and includes the stress stiffening effect. The mass matrix is also based on the updated geometry in this case. The stiffness matrix used in a free vibration analysis is also modified for the spin softening effect due to rotational loads when a large displacement analysis is requested. Therefore, the equation solved for a free vibration analysis with large displacement is

$$\{[K_L] + [K_{NL}] - \omega^2[M]\}[\Phi] = \{\lambda\}[M][\Phi] \quad (58)$$

where  $[K_L]$  is the linear stiffness matrix and  $[K_{NL}]$  is the stress stiffness (or nonlinear) matrix, and  $\omega$  is the rotational speed in radians per second.

The mass matrix is calculated element by element at the same time as the stiffness matrix. The equation for calculating the mass matrix for an element is

$$[M] = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} \rho [H_i]^T [H_j] |J| dr ds dt \quad (59)$$

which becomes a summation of the contribution of each integration point using numerical integration. These elemental mass matrices are saved and assembled into a system mass matrix.

The mass matrix used for the vibration analysis is a lumped mass matrix, which contributes to the economy of the problem without losing much accuracy. The solution schemes

include the capability to use a consistent mass matrix and the calculations to form elemental consistent mass matrices are also present in the code; however, the storage of these elemental matrices as well as the storage of the assembled consistent mass matrix has not been implemented.

In the lumped mass calculation, only the diagonal terms in the mass matrix integral equation are used. This results in a vector which contains the mass associated with each node. Each of the three degrees of freedom for a node has this same mass unless the degree of freedom is prescribed, in which case the associated mass for that degree of freedom is zero.

The eigenvectors represent the vibration modeshape and are normalized such that the maximum amplitude is 1.0. The kinetic and potential energies for each mode of vibration are calculated using the eigenvalues,  $\omega$ , normalized eigenvectors,  $\{\Phi\}$ , displacements,  $\{u\}$ , lumped mass matrix,  $[M]$ , and stiffness matrix,  $[K]$ , according to the equations

$$KE = \frac{\omega^2}{2} \{\Phi\}^T [M] \{\Phi\} \quad (60)$$

$$PE = \frac{1}{2} \{u\}^T [K] \{u\} \quad (61)$$

The modal mass is then calculated from the kinetic energy using the equation

$$m = \frac{2 KE}{\omega^2} \quad (62)$$

with the understanding that the normalized modeshapes represent a maximum deflection of 1.0 to arrive at the proper units of lb<sub>m</sub>.

The frequencies of vibration resulting from this analysis are printed on the results file (NT4) in CPS along with the potential energy, kinetic energy, and modal mass for each mode of vibration. The normalized mode shapes are printed on two files, NT68 and NT69. File NT69 contains only the normalized mode shapes in a node, dX, dY, dZ form for possible plotting purposes. File NT68, called a RAX file, contains the nodal masses as well as the frequencies and normalized mode shapes. This file can be used in calculating mode shape slopes, which are used in a flutter analysis.

A calculation of mode shape slope for blade vibration can be obtained on a limited basis for each vibration mode found. This calculation will be automatically initiated if a file with the



name SLOP (all uppercase letters) is found. The calculation of mode shape slope uses the equation

$$\frac{\phi b}{\delta} = \frac{\frac{1}{2} \sqrt{(\delta_{t_{LE}} - \delta_{t_{TE}})^2 + (\delta_{a_{LE}} - \delta_{a_{TE}})^2}}{\sqrt{(\delta_{t_{MC}})^2 + (\delta_{a_{MC}})^2}} \quad (63)$$

where  $\delta_t$  is tangential deflection,  $\delta_a$  is axial deflection, subscript *LE* refers to leading edge, subscript *TE* refers to trailing edge, and subscript *MC* refers to midchord.

### 3.1. 8. Buckling Analysis

Buckling analysis is very similar to a free vibration analysis. The generalized eigenproblem for buckling is

$$[K_L][\Phi] = \{\lambda\}[-K_{NL}]\{\Phi\} \quad (64)$$

where  $[K_L]$  is the linear stiffness matrix and  $[K_{NL}]$  is the stress stiffness (or nonlinear) matrix. Note that the negative of the stress stiffness is used. The eigenvectors in this case are normalized buckling shapes and the eigenvalues are multipliers to the current load that would give the critical load. This means that an eigenvalue of 1.0 indicates that the load is at the critical value. A negative eigenvalue indicates that buckling cannot occur in the current load configuration.

The inherent assumption in using the eigenvalues as load multipliers is that the load to stiffness relationship is linear. This may not be the case especially if the multiplier is large meaning that the applied load is a great deal less than the critical load for that distribution. To get a more accurate picture of the critical value for the load distribution it may be necessary to scale the load to the critical value (or closer to it) so that the linear assumption will be more accurate. Regardless, a positive eigenvalue greater than 1.0 indicates that buckling will not occur for the applied load, while a positive value less than 1.0 indicates buckling will occur for that load.

Buckling analysis can be turned on and off from load case to load case. Stiffnesses relating to the original configuration or to the current displaced configuration can be used allowing geometric nonlinearities to be taken into account.

### 3.1. 9. Micromechanics Analysis

Composite micromechanics analyses in the CSTEM program are performed using a modified version of the ICAN computer program obtained from NASA-Lewis. This program and the inputs are described in the ICAN User's Manual, which is included as an appendix. In the CSTEM application these inputs are obtained from the CSTEM input deck, are calculated from the results of the CSTEM finite element program, or are defaulted to a value such that a certain type of analysis is always performed.

#### 3.1. 9. 1. Data Bank of Constituent Properties

The ICAN materials data bank is assumed to exist as a file called 'ICANBNK' on the machine where the CSTEM program is being run. This file is opened by the CSTEM program (subroutine RDCAN) when the option for ICAN generation of material properties is activated.

#### 3.1. 9. 2. Calculation of Ply Material Properties

The ICAN routines can be accessed in two separate ways. This results from the fact that the ICAN program itself incorporates routines from a program called INHYD which calculates layer properties from the constituent fiber and matrix properties. These ICAN/INHYD type routines can be accessed to generate elastic material properties for composite systems whose constituent properties are contained in a data bank. Section XIV.3. in the CSTEM input sheets describe the input variables for this type of use.

The inputs needed to use ICAN to generate elastic properties are those contained in the Ply Details cards and Material System Details cards described in the ICAN User's Manual. The material identification number is assigned consecutively in order of input. All properties are returned in the local material coordinate system and are incorporated into the global system by the CSTEM program so that the orientation angle and thickness in the Ply Details card group are not necessary. The use temperature is passed from the CSTEM program to the ICAN/INHYD routines and is interpolated from nodal temperature values. There is a check for a trash temperature value

of 1234567. in which case a reference temperature of 70F is used. At present this would occur only if using the linear heat transfer analysis option without a corresponding structural analysis.

Material properties for a particular combination of unidirectional ply orientations can also be created. The individual unidirectional plies with the specified orientations are combined together into a laminate with ICAN. The laminate properties are then used as the material properties. This capability can be used to simulate 2D composite weave material properties.

### 3.1. 9. 3. Detailed Microanalyses

The use of ICAN as part of the CSTEM program for micromechanics analysis requires only that the material properties be generated by ICAN as previously described, that the location(s) where the analysis applies be specified, and whether strains/curvatures or stress resultants are to be calculated from the finite element results as input for the ICAN analyzer. The ICAN analysis is done immediately after the structural results have been computed and printed out so that an ICAN load case corresponds to a structural load case. However, an ICAN analysis can be done for up to a maximum number of different cross sections in a load case. This maximum is set by the parameter MXILC in the ICAN routines.

The cross sections within the structure where the ICAN analysis is to be performed are specified by inputting the surface element of a cross section. Section XXV. in the CSTEM input sheets describe these inputs. The cross section is then determined by finding the elements along the LAX axis from the surface element. The cross section elements should be lined up normal to the layers so that a true cross section is obtained. A maximum number of elements, defined by the parameter MAXIEL, can be in a cross section.

The calculations to obtain the necessary information for ICAN are done at user specified locations on the midsurfaces of the layers within the elements in the cross section. These specified locations are either the centroids, gauss points, or nodes of the cross section elements.

The number of layers in the cross section are determined by counting the number of layers in each consecutive element of the cross section starting from the bottom of the laminate (negative LAX). There is no check for layers extending between elements, so if a layer lies in two adjacent

elements it would be treated as two layers having the same orientation. The maximum number of layers through the cross section that can be handled is set by the parameter MAXPLY.

The analysis options set by the booleans described in the ICAN User's Manual are hardwired in the program as follows:

COMSAT	True
BIDE	True
CSANB	False
NONUDF	True

The boolean RINDV is determined by the CSTEM input variable ICAN.

RINDV is True if ICAN = 1 and False if ICAN = 2.

Much of the data required by ICAN as described in the Ply Details cards and Material System Details cards of the ICAN User's Manual is already available from the generation of the material properties using the ICAN / INHYD routines. The percent of moisture for each layer, which was input for the generation of material properties, is transferred to storage in array PL (row 72) for use by ICAN.

The use temperature for each layer is interpolated from the temperature of the element nodes to the midsurface of the layer using the element shape functions. ICAN uses the difference in this use temperature and the cure temperature of the material. This value is stored for each ply in array PL (row 50).

The thickness of each layer at the specified location is found as the fraction of the layer in the cross section thickness. The cross section thickness is found by summing element thicknesses calculated as projections on the surface normal of the scaled through thickness structural axis at the lower element face. This value is stored for each layer in array PL (row 7).

The orientation angle of the layer is obtained from the element layer information stored using the LAYIO routine of the CSTEM program. This angle is placed in the THLC vector used by the ICAN routines.

The bulk of the calculations done using the finite element results are to obtain the loadings for the ICAN analyzer. These calculations assume that the strain varies linearly through the cross

section, while the stress may be discontinuous. The loadings can be either strains and curvatures or loads in the form of stress resultants and couples. Strains and stresses at the cross section element integration points, along with the element geometry information are the quantities used from the finite element analysis. The integration points are generally in a Gauss distribution on each layer midsurface. In the case of an element with only one layer the integration points will be in a three dimensional Gauss distribution.

For the strain and curvature loading option of ICAN, the strains and curvatures at the cross section midsurface (reference plane) are needed. The strains are calculated by first determining the thickness through the entire cross section as the distance between the location on the upper surface of the cross section and the location on the lower surface. These locations are specified by input and their coordinates are calculated using shape functions and nodal coordinates. Once the cross section thickness has been determined the layer thicknesses are summed beginning from the bottom surface until this sum exceeds half of the cross section thickness. The strains at the integration points of the element in which the reference plane lies are then interpolated to the user specified location on the reference plane using Lagrange interpolation functions.

The reference plane curvature is calculated using the equation for strain from laminated plate theory which can be written as:

$$\{\epsilon\} = \{\epsilon^0\} - z\{\kappa^0\} \quad (65)$$

where  $\{\epsilon^0\}$  is the reference plane strain,  $\{\kappa^0\}$  is the reference plane curvature, and  $z$  is the distance from the reference plane to the location where the strain,  $\{\epsilon\}$ , is desired. This equation can be solved for the curvatures:

$$\{\kappa^0\} = (1/z) (\{\epsilon^0\} - \{\epsilon\}) \quad (66)$$

The curvatures can be calculated using this equation, the strain at the top layer of the cross section, and the distance from the reference plane to the specified location on the top layer.

The in plane stress resultants or membrane loads used by ICAN can be calculated using the equations

$$N_x = \int_{-h/2}^{+h/2} \sigma_x dz \quad (67)$$

$$N_y = \int_{-h/2}^{+h/2} \sigma_y dz \quad (68)$$

$$N_{xy} = \int_{-h/2}^{+h/2} \tau_{xy} dz \quad (69)$$

The equations for the stress couples or bending resultants are

$$M_x = \int_{-h/2}^{+h/2} \sigma_x z dz \quad (70)$$

$$M_y = \int_{-h/2}^{+h/2} \sigma_y z dz \quad (71)$$

$$M_{xy} = \int_{-h/2}^{+h/2} \tau_{xy} z dz \quad (72)$$

The equations for the transverse shear resultants are

$$Q_x = \int_{-h/2}^{+h/2} \tau_{xz} dz \quad (73)$$

$$Q_y = \int_{-h/2}^{+h/2} \tau_{yz} dz \quad (74)$$

In the previous equations,  $h$  is the cross section thickness and  $z$  is the distance from the reference plane.

Since the stress through the cross section is assumed to be piecewise linear these equations must be integrated piecewise, integrating over each individual layer and summing the layer results over the cross section. For the membrane loads this can be written in general as

$$\{N\} = \sum_1^L \int_{z_r^-}^{z_r^+} \{\sigma\} dz \quad (75)$$

where  $z_r$  is the distance from the reference plane to the layer midplane and the summation limit,  $L$ , is the total number of layers in the cross section. The upper integral limit,  $z_r^+$ , can also be written as  $z_r + h/2$  and is the distance from the reference plane to the upper surface of the layer being integrated. The lower integral limit,  $z_r^-$ , is similarly  $z_r - h/2$ . Assuming the stress to be linear over the layer

$$\{\sigma\} = \{\sigma_{mid}\} + \frac{z_l}{h_l} (\{\sigma_{top}\} - \{\sigma_{bot}\}) \quad (76)$$

where  $\{\sigma_{mid}\}$  is the layer midsurface stress,  $\{\sigma_{top}\}$  is the stress at the layer top surface,  $\{\sigma_{bot}\}$  is the stress at the layer bottom surface, and  $z_l$  is the distance from the layer midsurface to a point in the layer and can also be written as

$$z_l = z - z_r \quad (77)$$

Substituting this stress function into the integral gives

$$\{N\} = \sum_1^L \left\{ \{\sigma_{mid}\} \int_{z_r^-}^{z_r^+} dz + \frac{\{\Delta\sigma\}}{h} \int_{z_r^-}^{z_r^+} z dz - \frac{\{\Delta\sigma\}}{h} z_r \int_{z_r^-}^{z_r^+} dz \right\} \quad (78)$$

where  $\{\Delta\sigma\} = \{\sigma_{top}\} - \{\sigma_{bot}\}$

When the previous equation is integrated, the resulting general equation for the membrane loads is

$$\{N\} = \sum_1^L \{\sigma_{mid}\} h \quad (79)$$

The transverse shear resultant integrals are of the same form as those for the membrane loads. The bending resultants are somewhat different and use of the same procedure results in a general equation for the bending resultants:

$$\{M\} = \sum_1^L \{\sigma_{mid}\} z_r h + \frac{\{\Delta\sigma\}}{12} h^2 \quad (80)$$

The quantities needed in the equations for the loads are obtained by looping through all the layers beginning from the bottom of the cross section, calculating the contribution of that layer to the loads and summing. The stresses at the middle, top, and bottom surface locations are calculated from the strains at these same locations. The strains are interpolated from all the integration points within the element using Lagrange interpolation functions. Temperatures at the middle, top, and bottom surface locations are interpolated from the element nodes using the element shape functions. The material matrices are then calculated so that the stress at these points can be obtained from the strains.

### 3.1. 10. Enhanced Shear Stress Recovery

Calculation of interlaminar shear stresses for layered finite elements presents a particular difficulty due to the discontinuous nature of these stresses at the layer interfaces. The main



difficulty is that of determining (or assuming) which components of stress and strain are continuous across the interface. Assuming that the interface remains perfectly bonded requires that the displacements  $u$ ,  $v$  and  $w$  must all be continuous across the interface. The usual finite element method of stress recovery makes use of the derivatives of the element shape functions to calculate strains, then calculates stresses from these strains. This implies continuity of strain across layer interfaces.

Examination of equilibrium considerations at a layer interface requires that stresses acting out of the plane of the layers (across the interface) must be continuous. This means that in plane strains and out of plane stresses are the quantities which remain continuous across layer interfaces, which is in contradiction to the usual finite element stress recovery method. The out of plane stresses calculated by the finite element method will be discontinuous, and generally represent an average value through the layer.

For this reason, there is an enhanced interlaminar shear stress recovery procedure available in the CSTEM program which allows an alternate method of calculating interlaminar shear stresses to be used. This method makes use of the stress equilibrium equations and the in plane stresses calculated using the usual finite element stress recovery method.

The differential equations of equilibrium are arrived at by considering a small rectangular volume under a general state of stress. Summing forces due to the stress components acting over the differential faces of the volume results in the following equations of equilibrium:

$$\begin{aligned}
 \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + X &= 0 \\
 \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yz}}{\partial z} + Y &= 0 \\
 \frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + Z &= 0
 \end{aligned}
 \tag{81}$$

where  $X$ ,  $Y$ , and  $Z$  represent forces acting on the volume. Looking at the second of these equations and ignoring the body force,  $\tau_{yz}$  can be solved for :

$$\tau_{yz} = \int \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} dz \quad (82)$$

If the layers in the finite element are thin, the stress components on the right side of the above equation can be assumed to vary linearly through the layer thickness. Since the in plane strain components are continuous as assumed by the finite element method, the in plane stress components calculated using the usual finite element stress recovery method are accurate. In the CSTEM program these stress values are calculated at integration points, which are located at the layer midsurfaces. The in plane stress gradients can be approximated by interpolating the in plane stresses to some desired location and to points which are a small  $x$  and  $y$  distance from the desired location. By using a simple rectangle rule, equation (82) can then be integrated layer by layer to give the desired enhanced interlaminar shear stress result.

Equation (83) illustrates this method, where  $h_l$  is the layer thickness,  $\Delta x$  and  $\Delta y$  are the

$$(\tau_{yz})_{n+1} = (\tau_{yz})_n + h_l \left[ \frac{\sigma_y - \sigma_{y\Delta y}}{\Delta y} + \frac{\tau_{xy} - \tau_{xy\Delta x}}{\Delta x} \right] \quad (83)$$

small  $x$  and  $y$  distances from the desired location, and  $n$  represents the layer number. If no traction loads are applied,  $\tau_{yz}$  is zero at the top of the first ply (i.e.  $(\tau_{yz})_0 = 0$ ). A partial sum of these quantities (from a free face, down to an interface) is equal to the shear stress acting on that interface. The  $\tau_{xz}$  stress component can be found similarly.

An additional feature of this method which can be applied makes use of the fact that the usual finite element method of stress recovery will result in some average out of plane stress that accurately predicts the shear stress resultant within the element. This resultant can be used to scale the enhanced shear stress profile to give the same shear stress resultant as determined using the usual finite element stress recovery method.

### 3.1. 11. Nonlinear Material Analysis

The capability to perform several types of nonlinear material structural analyses exists in CSTEM. These include plasticity, damage, and creep analyses. Several different models to describe these types of nonlinear material behavior are available. The different models are generally applied to a particular material type, so that several models may be included in a single structural analysis which contains several material types, or a single model may be applied to several material types as desired by the user. Some combinations of the different models may not be appropriate if the input constants supplied by the user have been generated independently with no consideration of interaction between the phenomena which the various models attempt to predict. This may be particularly true with respect to the combination of damage with plasticity and/or creep. If used together, the sequence is to perform the plasticity analysis first, then the damage analysis, then the creep analysis.

#### 3.1. 11. 1. Iteration Methods

All nonlinear material analyses in CSTEM are iterative in nature. The iteration methods used in CSTEM are one of two types: initial stiffness (right hand side) or tangent stiffness. The following descriptions are based on plasticity iteration, but are similar for damage and creep. Creep models are only available using the initial stiffness method of iteration.

##### 3.1. 11. 1.1. Initial Stiffness Method

The initial stiffness iteration methods are also known as right hand side or initial stress methods. With this method, the initial elastic stiffness matrix is used throughout the iteration process. The extra displacements due to the inelastic strains are accounted for by the use of inelastic pseudo forces applied to the model. There are two slightly different initial stiffness methods which can be chosen by the CSTEM user as described in Section 4.1.1.3.4. The first calculates the pseudo forces as the residual forces resulting from the imbalance between applied forces and internal forces due to the stresses. The second method, which is the default, calculates

the pseudo forces directly from the inelastic strains. Figure 11. charts the initial stiffness procedure with the pseudo force calculated directly from the inelastic strains.

Prior to beginning the iteration process, elastic displacements for the new load condition are calculated assuming purely linear material behavior from the previous load condition. Any nonlinear material behavior which occurred in a previous load condition is included by the application of the pseudo forces calculated from the previous nonlinear strains.

For the first iteration, the current inelastic strain increment and pseudo force increment are zero, ( $\{\delta\epsilon_c\} = 0$  and  $\{\delta F_{inelas}\} = 0$ ). The following steps are performed at each integration point within every element.

The total strain is calculated from the total displacements as

$$\{\epsilon_{tot}\}^i = [B]^T \left( \{d\} + \{\delta d\}^{i-1} \right) \quad (84)$$

Elastic strains are calculated by subtracting any thermal strains, previous inelastic strains, and current inelastic strain increments. Stresses are calculated from these elastic strains:

$$\{\sigma\}^i = [D] \left\{ \{\epsilon_{tot}\}^i - \{\epsilon_{therm}\} - \{\epsilon_{inelas}\} - \{\delta\epsilon_{inelas}\}^{i-1} \right\} \quad (85)$$

For orthotropic materials the stresses, strains, and material matrix are transformed into the local material coordinate system. For a layered fibrous composite these quantities are oriented with 11 in the fiber direction, 22 in the layer plane transverse to the fibers, and 33 out of the layer material plane. Material properties (such as stress-strain curves, creep coefficients, damage coefficients) are interpolated to the temperature of the integration point. The inelastic material model is then entered which calculates an inelastic strain increment. Convergence at the integration point is checked by comparison to incremental values from the previous iteration. The newly calculated inelastic strain increments are stored and the contribution of the integration point to the inelastic pseudo force is then calculated. The incremental pseudo force calculation represents the primary difference between the two different initial stiffness methods in CSTEM.

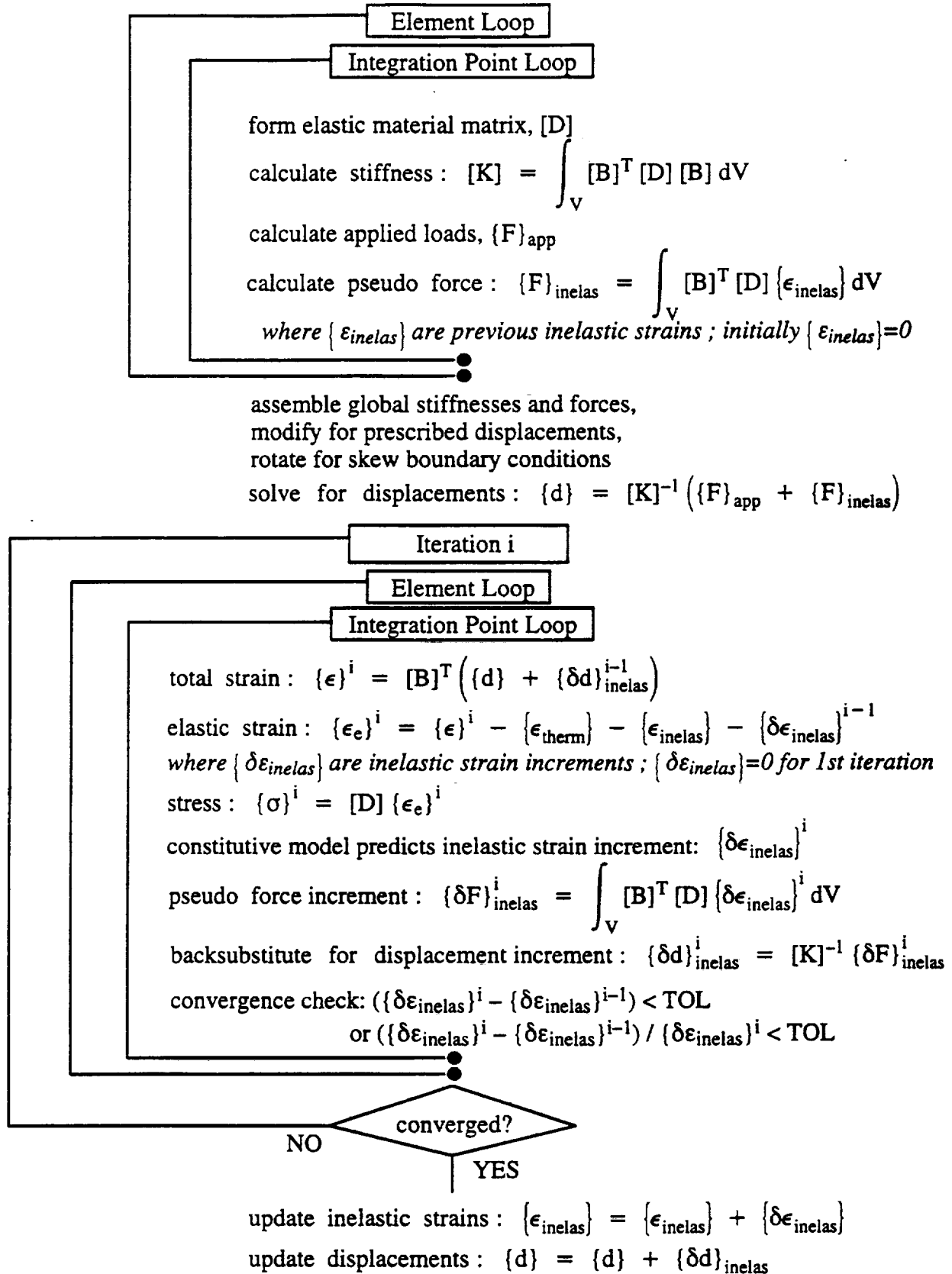


FIGURE 11. INITIAL STIFFNESS FLOWCHART

The first method calculates the incremental pseudo force as the residual force resulting from the imbalance between applied forces and internal forces due to the stresses:

$$\{\delta F_{\text{inelas}}\}^i = \{F_{\text{app}}\} - \int_V [B_L]^T \{\sigma\} dV \quad (86)$$

where  $\{F_{\text{app}}\}$  is the applied force.

In the second method, the incremental pseudo force is determined directly from the incremental inelastic strains as:

$$\{\delta F_{\text{inelas}}\}^i = \int_V [B]^T [D] \{\delta \epsilon_{\text{inelas}}\}^i dV \quad (87)$$

Once the contributions from all elements have been calculated, the elemental pseudo forces are assembled into a global pseudo force and incremental displacements are determined by backsubstituting the incremental pseudo force into the decomposed initial stiffness. If convergence at all integration points was not achieved, another iteration is begun.

### 3.1. 11. 1.2. Tangent Stiffness Method

The tangent stiffness method reforms the stiffness matrix to account for the increased flexibility in the structure due to the occurrence of inelastic strains. This is purely an incremental method, with even linear elastic solutions performed in an incremental manner. The default tangent stiffness method implemented in CSTEM is a modified Newton–Raphson method. The stiffnesses are not reformed with each inelastic iteration, but are updated at the beginning of each new load condition only. The user can change the stiffness recalculation frequency by using the NRIT keyword option as described in Section 4.1.1.3.5. During the iteration process, the inelastic incremental displacements are accounted for by incremental pseudo forces in a manner similar to the initial stiffness method. The Sun orthotropic plasticity model and the ceramic matrix composite damage model, CMCUMAT, can be used with the tangent stiffness method as well as with the initial stiffness method. Stiffness calculation can be time consuming, so elements which do not have additional inelastic strains occurring in the load increment will not have stiffnesses reformed. Of course, if there are conditions in the next load condition which would require reforming the

stiffness (such as changes in temperature or material) then the stiffness will be reformed as usual. For load conditions which cause elastic unloading, the stiffness of unloading integration points which were previously inelastic will be recalculated during the iteration process using the initial stiffness since the tangent stiffness is always used at the beginning of a load condition when reforming the stiffness.

### 3.1. 11. 2. Plasticity Analysis

The plasticity models in CSTEM predict time independent nonlinear behavior for materials subject to cyclic loadings and large, nonuniform excursions in temperature. There are currently two isotropic material plasticity models and one orthotropic material plasticity model included in CSTEM. The isotropic material models are a kinematic hardening plasticity model and an isotropic hardening plasticity model. The orthotropic material model is the C.T. Sun plasticity model. A user interface (CHOOK) for user defined plasticity models is also available.

#### 3.1. 11. 2.1. Isotropic Material with Kinematic Hardening

This model utilizes the McKnight modification of the subvolume method originally proposed by Besseling. It assumes that each volume of material consists of different subvolumes of elastic – perfectly plastic material with the same elastic properties but different yield strengths. The total strain is the same in each subvolume, so the difference between subvolumes occurs in the proportion of elastic and plastic strains in each. The combined stress–strain curve for the full volume of material decreases in slope as the stress reaches the yield stress for a subvolume and that subvolume ceases to carry additional load. The mathematical model thus produces a piecewise linear, strain hardening stress–strain curve. The Bauschinger effect (subsequent yield stress after yielding in the opposite loading direction) is modelled as kinematic hardening, which assumes that reyielding will occur after traversing a stress range of two times the initial yield stress. If viewed in terms of a yield surface, the yield surface remains a constant size but moves due to yielding and inelastic action. Figure 12. illustrates the Bauschinger effect. As implemented in CSTEM, this model works in terms of strain rather than stress.

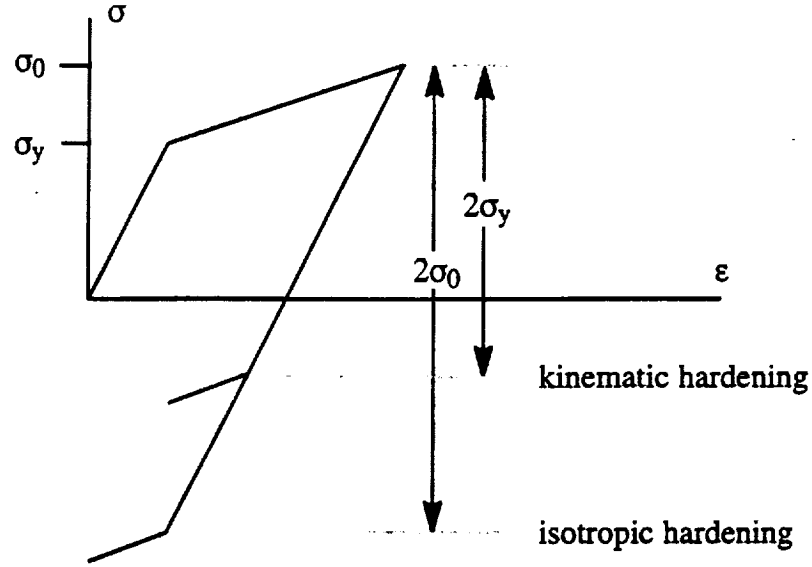


FIGURE 12. BAUSCHINGER EFFECT THEORIES

In order to utilize uniaxial stress-strain data, scalar quantities obtained from multiaxial stress and strain tensors are used throughout the plasticity model. These scalar quantities are called effective stresses and strains. The uniaxial stress-strain data can be converted to effective stress-strain data and used directly with the calculated effective stress and effective strain states to determine the effective plastic strain increment. Essentially, the effective stress is checked against the yield stress using the Von Mises yield criterion. If yielded, the effective plastic strain increment is determined from the uniaxial effective stress-strain data. The components of the plastic strain increment tensor are determined by using a flow rule associated with the Von Mises yield criterion called the Prandtl-Reuss flow rule, which relates the components of the plastic strain increment with the current stress components.

The Von Mises yield criterion, used to define the onset of inelastic behavior, can be written

$$\sigma_{yield}^2 = \frac{1}{2} \left[ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2) \right] \quad (88)$$

This leads to the definition of the scalar quantity called effective stress.

$$\sigma_{eff} = \frac{1}{\sqrt{2}} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2)} \quad (89)$$



The Prandtl–Reuss flow rule requires that the plastic strain increment at any instant of loading be proportional to the instantaneous deviatoric stress (i.e. normal to the yield surface). This can be written in terms of stress and strain components as

$$\begin{aligned}\delta\epsilon_x^p &= \frac{2}{3} \delta\lambda \left[ \sigma_x - \frac{1}{2}(\sigma_y + \sigma_z) \right] \\ \delta\epsilon_y^p &= \frac{2}{3} \delta\lambda \left[ \sigma_y - \frac{1}{2}(\sigma_z + \sigma_x) \right] \\ \delta\epsilon_z^p &= \frac{2}{3} \delta\lambda \left[ \sigma_z - \frac{1}{2}(\sigma_x + \sigma_y) \right]\end{aligned}\tag{90}$$

$$\delta\epsilon_{xy}^p = \delta\lambda \sigma_{xy} ; \delta\epsilon_{yz}^p = \delta\lambda \sigma_{yz} ; \delta\epsilon_{zx}^p = \delta\lambda \sigma_{zx}$$

The proportionality constant,  $\delta\lambda$ , can be derived by making use of the yield criterion giving

$$\delta\lambda = \frac{3}{2} \frac{\delta\epsilon_p}{\sigma_{eff}}\tag{91}$$

where the definition for effective strain is used to give the effective plastic strain increment as

$$\delta\epsilon_p = \frac{\sqrt{2}}{3} \sqrt{(\delta\epsilon_x^p - \delta\epsilon_y^p)^2 + (\delta\epsilon_y^p - \delta\epsilon_z^p)^2 + (\delta\epsilon_z^p - \delta\epsilon_x^p)^2 + 6(\delta\epsilon_{xy}^p{}^2 + \delta\epsilon_{yz}^p{}^2 + \delta\epsilon_{zx}^p{}^2)}\tag{92}$$

A more detailed explanation can be found in the book by Alexander Mendelson titled Plasticity: Theory and Application published by the MacMillan Company. The subvolume method as used in CSTEM is described in the Ph.D. dissertation of Richard McKnight titled Finite Element Cyclic Thermoplasticity Analysis by the Method of Subvolumes, University of Cincinnati, 1975.

### 3.1. 11. 2.2. Isotropic Material with Isotropic Hardening

This model is a simple stress based plasticity model, utilizing the input stress–strain curve in effective form and the current effective stress state to determine the plastic strain. This model does not perform well for reverse cycling plasticity problems, and its use in such cases should be avoided. It should essentially be used for analysis of structures with yielding and subsequent elastic cycling only or continued yielding in the initial yield direction (tension or compression).

This model utilizes the Von Mises yield criterion, definition of effective stress, and Prandtl–Reuss flow rule (equations (88), (89), and (90)), similar to the kinematic hardening plasticity model.

### 3.1. 11. 2.3. Orthotropic Material

This model uses the method of C.T Sun to predict 3D plasticity. It is based on the assumption that there is no plasticity caused by uniform dilatation as opposed to the usual assumption in the isotropic models that hydrostatic stress causes no plasticity. This was verified by finite element analysis of a representative volume of fibers in a matrix in which the constituents can be modelled using isotropic material plasticity. Loading the model in a hydrostatic stress state ( $\sigma_{11}=\sigma_{22}=\sigma_{33}=\sigma$ ) results in nonlinear response in the transverse normal strains with stress, while loading the model in a uniform dilatational state ( $\epsilon_{11}=\epsilon_{22}=\epsilon_{33}=\epsilon$ ) results in a linear stress–strain response. Based on this assumption, Sun proposed a plastic potential function of the form:

$$2f(\sigma_{ij}) = A_{12}(\sigma_{11} - \alpha_{12}\sigma_{22})^2 + A_{23}(\sigma_{22} - \alpha_{23}\sigma_{33})^2 + A_{13}(\sigma_{33} - \alpha_{31}\sigma_{11})^2 + 2a_{44}\sigma_{23}^2 + 2a_{55}\sigma_{13}^2 + 2a_{66}\sigma_{12}^2 \quad (93)$$

where the constants  $A_{ij}$  and  $\alpha_{ii}$  are determined from material testing.  $A_{23} = 1$  is generally used. If there is no plasticity in the fiber direction, ( $d\epsilon_{11}^p = 0$ ), then  $A_{12} = A_{13} = 0$ . If the material is transversely isotropic then  $\alpha_{23} = 1$  and  $\alpha_{44} = \alpha_{55}$ .

The coefficients  $\alpha_{ij}$  are based on the elastic constants of the material so that the initial plastic anisotropy is influenced by the elastic anisotropy of the material. These coefficients are defined as

$$\begin{aligned} \alpha_{12} &= \frac{C_{11} + C_{12} + C_{13}}{C_{12} + C_{22} + C_{23}} \\ \alpha_{23} &= \frac{C_{12} + C_{22} + C_{23}}{C_{13} + C_{23} + C_{33}} \\ \alpha_{31} &= \frac{C_{13} + C_{23} + C_{33}}{C_{11} + C_{12} + C_{13}} \end{aligned} \quad (94)$$

As implemented in CSTEM, these coefficients can be artificially set to any value or allowed to assume the defined values based on the elastic constants. Setting these coefficients all to a value of 1 results in Hill's theory of plasticity for orthotropic materials.

The associated flow rule for the potential function of equation (93) is

$$d\epsilon_{ij}^p = d\lambda \frac{\partial f}{\partial \sigma_{ij}} \quad (95)$$

The effective stress is defined as  $\bar{\sigma} = \sqrt{3f}$ . Substituting equation (93) gives

$$\bar{\sigma}^2 = \frac{3}{2} [ A_{12}(\sigma_{11} - \alpha_{12}\sigma_{22})^2 + A_{23}(\sigma_{22} - \alpha_{23}\sigma_{33})^2 + A_{13}(\sigma_{33} - \alpha_{31}\sigma_{11})^2 + 2a_{44}\sigma_{23}^2 + 2a_{55}\sigma_{13}^2 + 2a_{66}\sigma_{12}^2 ] \quad (96)$$

By differentiating the effective stress, the effective stress increment can be written as

$$\begin{aligned} d\bar{\sigma} = \frac{3}{2\bar{\sigma}} \{ & [ A_{12}(\sigma_{11} - \alpha_{12}\sigma_{22}) - A_{13}\alpha_{31}(\sigma_{33} - \alpha_{31}\sigma_{11}) ] d\sigma_{11} + \\ & [ A_{23}(\sigma_{22} - \alpha_{23}\sigma_{33}) - A_{12}\alpha_{12}(\sigma_{11} - \alpha_{12}\sigma_{22}) ] d\sigma_{22} + \\ & [ A_{13}(\sigma_{33} - \alpha_{31}\sigma_{11}) - A_{23}\alpha_{23}(\sigma_{22} - \alpha_{23}\sigma_{33}) ] d\sigma_{33} + \\ & [ 2a_{44}\sigma_{23} ] d\sigma_{23} + [ 2a_{55}\sigma_{13} ] d\sigma_{13} + [ 2a_{66}\sigma_{12} ] d\sigma_{12} \} \end{aligned} \quad (97)$$

When the associated flow rule is combined with the definition of effective stress and the concept of plastic work, the proportionality factor,  $d\lambda$ , can be obtained as

$$d\lambda = \frac{3}{2} \frac{d\epsilon^p}{\bar{\sigma}} = \frac{3}{2H_p\bar{\sigma}} d\bar{\sigma} \quad (98)$$

where the plastic modulus,  $H_p = \frac{d\bar{\sigma}}{d\epsilon^p}$ , is obtained from the effective stress – effective plastic strain curve obtained from material testing.

Material testing to obtain the necessary constants involve tension tests of off-axis specimens. The material potential function constants of equation (93) determined from these tests must be chosen so that the resulting effective stress – effective plastic strain curve is independent of the in plane orientation of the material. Using equation (96), definition of the potential function constants allows calculation of the effective stress from the uniaxial test stress component. The

effective plastic strain can then be obtained from the flow rule (equation (95)), giving

$$\begin{aligned}
 (\bar{\epsilon}^p)^2 = & -\frac{2}{3} \left[ \frac{(A_{13} + A_{23}a_{23}^2)\epsilon_{11}^p \epsilon_{22}^p + (A_{12}a_{12}a_{23} + A_{13}a_{31})\epsilon_{22}^p \epsilon_{33}^p + (A_{23}a_{23} + A_{12}a_{12}^2a_{23})\epsilon_{11}^p \epsilon_{33}^p}{A_{12}A_{13}a_{12} + A_{12}A_{23}a_{12}a_{23}^2 + A_{13}A_{23}a_{31}a_{23}} \right] \\
 & + \frac{2}{3} \left[ \frac{(\gamma_{23}^p)^2}{a_{44}} + \frac{(\gamma_{13}^p)^2}{a_{55}} + \frac{(\gamma_{12}^p)^2}{a_{66}} \right] \quad (99)
 \end{aligned}$$

Referring to the equations listed above, the steps involved in calculating an increment of plastic strain using the orthotropic plasticity model as implemented in CSTEM are listed below.

- Calculate stress from elastic strain.
- Stress increment obtained  
by subtracting previous load case stress from current load case stress.
- Interpolate material constants for temperature ( $A_{12}$ ,  $A_{23}$ ,  $A_{13}$ ,  $a_{44}$ ,  $a_{55}$ ,  $a_{66}$ )  
( $a_{12}$ ,  $a_{23}$ ,  $a_{31}$  may be calculated from elastic constants or interpolated from input).
- Calculate effective stress,  $\bar{\sigma}$ , using interpolated constants and current stress. (96)
- Calculate effective stress increment,  $d\bar{\sigma}$ , using interpolated constants, current stress, current effective stress, and stress increment. (97)
- Interpolate master effective stress – effective plastic strain curve for temperature.
- From curve, determine effective plastic strain corresponding to current effective stress (yielding occurs if effective plastic strain > 0).
- Determine plastic modulus as differential of power law curve or as slope of portion of piecewise linear curve containing current effective stress.
- Calculate proportionality factor,  $d\lambda$ , using current effective stress, effective stress increment, and plastic modulus. (98)
- Calculate plastic strain increments from flow rule using  $d\lambda$ , interpolated material constants, and current stress. (95)
- Calculate effective plastic strain increment using  $d\lambda$ , and current effective stress. (98)

These steps are performed for each integration point. Iteration continues until convergence is achieved.

A more detailed description of this plasticity model can be found in "A Plastic Potential Function Suitable for Anisotropic Fiber Composites", J.L. Chen and C.T. Sun, Journal of Composite Materials, Vol. 27, No. 14, 1993.

#### 3.1. 11. 2.4. CHOOK User Plasticity

A method is available by which a user's custom plasticity model can be included for use in CSTEM. User plasticity requires the user to compile the user plasticity model with the CSTEM code. The user plasticity model receives information from CSTEM and returns information to CSTEM through a single routine called CHOOK, which links the user plasticity model into the CSTEM iteration scheme. The calling arguments of the CHOOK routine contain vectors which are sized by user specification of certain input deck variables. These vectors contain some predefined quantities, such as stress, strain, temperature, etc., as well as user defined information about the current state of the material. The user updates and stores whatever internally generated user defined information is needed by the model in these vectors within the vector lengths defined by the input deck variables. The vectors are returned to CSTEM and stored on a random access file for each calculation location. Calculations are performed at each integration point in the finite element model. Figure 13. shows the CHOOK calling sequence and the predefined vector locations. User model data is stored after the predefined data locations. Section 4.1.12.3.3. describes user plasticity inputs as entered on the input deck.

The predefined parameters in the integer properties vector, IPROPS, are all defined by CSTEM with the exception of IPROPS(5) and IPROPS(6). The structural/heat transfer flag is 0 for structural, 1 for heat transfer. For user plasticity, this will always indicate a structural solution.. The material number, iteration number, integration point number, element number, and cycle number are all fairly self explanatory. The debug flag can be set by the plasticity debug keyword input. The units flag will always indicate 0.

# SUBROUTINE CHOOK

(IPROPS, LIPS, RPROPS, LRPS, EPSI, LEPS, STATP0, STATP, LSTP, STATM0, STATM, LSTM, D)

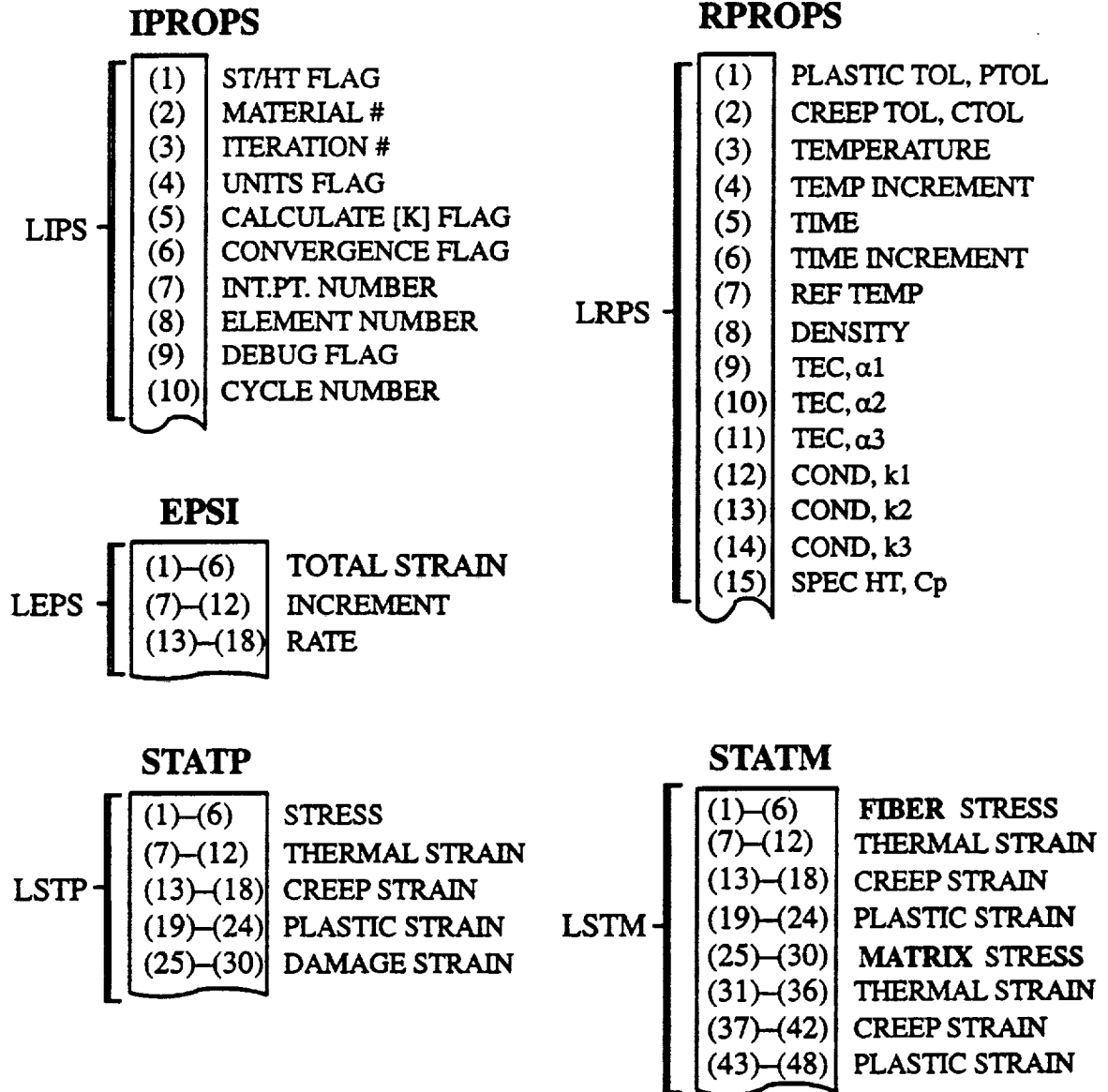


FIGURE 13. USER PLASTICITY INTERFACE SUBROUTINE CHOOK

The user should return appropriate values for IPROPS(5), and IPROPS(6). IPROPS(5) contains a flag for use with the tangent stiffness iteration technique. IPROPS(5)=0 indicates not to recalculate the element stiffness based solely on user plasticity results, IPROPS(5)=1 indicates recalculation of the element stiffness should occur regardless of any other considerations. The choice of modified or full Newton–Raphson iteration techniques determines when this recalculation occurs. IPROPS(6) is a flag indicating convergence. IPROPS(6)=0 indicates no plasticity occurred (elastic only), IPROPS(6)=1 indicates converged plasticity, IPROPS(6)=2 indicates unconverged plasticity. For use with the tangent stiffness technique, IPROPS(6)=-1 indicates elastic unloading. CSTEM will perform convergence checking if IPROPS(6)=100 is returned. CSTEM performs convergence checking using a default value and technique, which is to declare convergence when the change in Von Mises effective plastic strain increment between iterations is less than  $1.E-5$  inches/inch at each integration point.

The variables in the real properties vector, RPROPS, are all defined by CSTEM for use in the user subroutine. The plastic convergence tolerance, RPROPS(1), and creep convergence tolerance, RPROPS(2), are values input by the user. The temperature, RPROPS(3), is the current temperature, after application of the temperature increment, RPROPS(4). The temperature prior to the temperature increment is obtained by subtracting RPROPS(4) from RPROPS(3). The same is true for the time, RPROPS(5), and time increment, RPROPS(6). The properties are at the temperature contained in RPROPS(3).

The variables in the strain vector, EPSI, are the current total strain, the current total strain increment, and the current total strain rate. To obtain the total strain at the previous converged solution point, subtract the total strain increment from the total strain stored in EPSI. The total strain rate is simply the total strain increment divided by the time increment.

The variables in the ply level state vector, STATP, are the values due to the previous iteration. The stress is calculated from current displacements, the thermal strain is that due to the current temperature, and the creep, plastic, and damage strains are those obtained from the user models or other models as appropriate. The stress and plastic strain should be updated by the user for the current iteration. The related vector, STATP0, contains the last converged values of the

same quantities as in STATP. When convergence is achieved, the entire current STATP vector is saved in STATP0, including any user quantities which follow the predefined variables. In this way, the previously converged values are available for use in the user model in a subsequent load case.

The variables in the micro state vector, STATM, are not generated by CSTEM. They are to be used as storage by the user routine. These quantities are predefined so that they may be used for output purposes. Currently, there is no access to these quantities for output. The vector STATM0 is similar to STATP0 in that it contains the previously converged micro state variables.

User data specific to material type can be entered from the input deck. It is accessible for use in a user material model through the routine UMATIO. The material data is stored in the same order as read from the input deck. The length of the vector containing data for a material is specified by the input variable MATQ, which may be different for each material.

## SUBROUTINE UMATIO (IMAT, MATQ, V, IW)

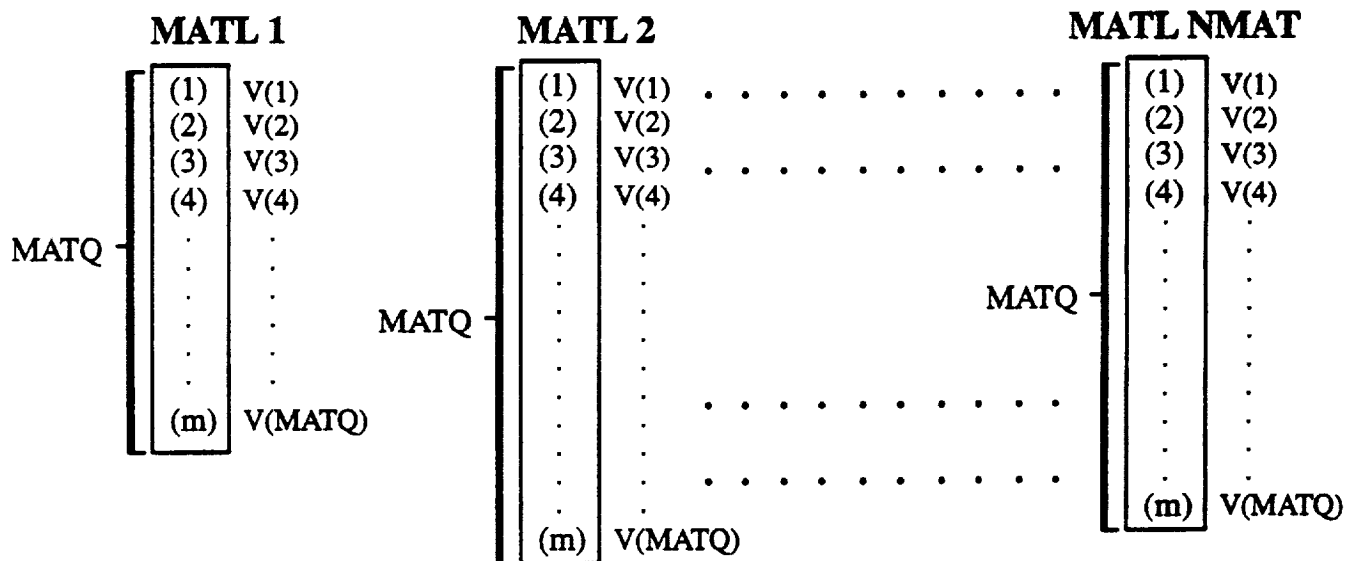


FIGURE 14. USER MATERIAL DATA INTERFACE ROUTINE UMATIO



### 3.1. 11. 3. Damage Analysis

There is one damage model implemented in CSTEM. This model is due to Leckie et.al. and was developed for use in CSTEM with funding from the Enabling Propulsion Materials contract of the High Speed Civil Transport program. There is also a simple stiffness degradation method as well as a method for implementing a user defined damage model.

Damage mechanics analyses may be performed using either an initial stiffness or a tangent stiffness method (see Section 3.1. 11. 1.). An acceleration calculation is utilized for the pseudo force initial stiffness method. This acceleration calculation can be modified or deactivated by the user as described in Section 4.1.1.3.6. The acceleration calculations have resulted in faster convergence of various damage loadings. In some cases, acceleration has also resulted in non-convergence of load cases which do converge with acceleration turned off. If an unconverged load case is encountered, the user should first attempt to restart the analysis from the last converged load case, changing the acceleration iteration interval or turning acceleration on or off as appropriate relative to the unconverged load case before resorting to a reduction of the load increment.

#### 3.1. 11. 3.1. CMCUMAT Ceramic Matrix Composite Damage Model

The damage model implemented in CSTEM is due to Leckie, et.al. This damage model is for ceramic matrix composites and models matrix cracking, debonding, fiber breakage, and fiber pull out. These different mechanisms produce a loss of stiffness and inelastic strains. The general procedure is based on continuum damage mechanics and is applied in the CSTEM implementation to a [0/90] laminate composite. Necessary material constants are obtained using the results of pure tension tests of two laminate orientations, with each test involving a series of loading and unloading sequences.

The CMCUMAT model is continuing to be developed. It is probably best, therefore, to simply refer the reader to the latest documentation on the model, which is a currently unpublished paper from the University of California, Santa Barbara Department of Mechanical and

Environmental Engineering titled "Continuum Description of Damage in Ceramic Matrix Composites" by Burr, Hild, and Leckie.

### 3.1. 11. 3.2. Polynomial Stiffness Degradation

The polynomial stiffness degradation model can be used as a simple method to change the in plane elastic moduli (E1, E2) as a function of stress and cycles. All other elastic material constants remain unchanged. This change is calculated according to the equation:

$$\frac{E}{E_0} = 1 + \{A \ln(N) + B \ln(N^2) + C \ln(N^3) + D \ln(N^4)\} * \left\{ S_A + S_B \left( \frac{\sigma}{S_U} \right) + S_C \left( \frac{\sigma}{S_U} \right)^2 \right\} \quad (1)$$

where the constants A, B, C, D, S<sub>A</sub>, S<sub>B</sub>, S<sub>C</sub>, and S<sub>U</sub> are input material constants, N is the cycle number, and σ is the stress component corresponding to the modulus being considered.

### 3.1. 11. 3.3. DHOOK User Damage

A method is available by which a user's custom damage model can be included for use in CSTEM. User damage requires the user to compile the user damage model with the CSTEM code. The user damage model receives information from CSTEM and returns information to CSTEM through a single routine called DHOOK, which links the user damage model into the CSTEM iteration scheme. The DHOOK routine is identical to the user plasticity routine CHOOK, except that it is accessed from the damage analysis iteration scheme rather than the plasticity analysis iteration scheme. The calling arguments of the DHOOK routine contain vectors which are sized by user specification of certain input deck variables. These vectors contain some predefined quantities, such as stress, strain, temperature, etc., as well as user defined information about the current state of the material. The user updates and stores whatever internally generated user defined information is needed by the model in these vectors within the vector lengths defined by the input deck variables. The vectors are returned to CSTEM and stored on a random access file for each calculation location. Calculations are performed at each integration point in the finite element model. Figure 13. shows the CHOOK calling sequence and predefined vector locations, which are identical to the DHOOK calling sequence and predefined vector locations. User data is

stored after the predefined data locations. The UMATIO routine concept shown in Figure 14. can be utilized by the user damage model for access and storage of user material data.

#### 3.1. 11. 4. Creep Analysis

Creep is time dependent plastic flow of a material. The creep response of a material can typically be divided into three regions: primary, secondary, and tertiary. During primary creep the material undergoes rapid inelastic response. During secondary creep the material undergoes a nearly linear inelastic response. Tertiary creep is another region of rapid inelastic response which occurs prior to rupture. The creep models in CSTEM model primary and secondary creep regions.

There is currently an isotropic material creep model and an orthotropic material creep model in CSTEM. In addition, there is a method available for the implementation of a user defined creep model. Creep analyses are performed after any plasticity and damage analyses.

##### 3.1. 11. 4.1. Time Incrementing

Creep occurs over a period of time (a time step) in which the loads and temperatures on the model remain constant (a load case). The time step is input for the load case with a method to subincrement this time step selected by the user. There are two time subincrement methods available.

The first method is to use a defined number of equivalent time subincrements. This is a fairly straightforward method. The user specifies how many subincrements are to be used in the time step and each subincrement will be  $\delta t = \Delta t / N$  long for N subincrements. In order to converge, the time steps in primary creep must generally be much smaller than the time steps in secondary creep. This method requires the user to break up the creep time into steps that allow reasonable time subincrements for the amount of creep experienced. If a time subincrement is too large, convergence will not be achieved. If a time subincrement is too small, the analysis will take an unnecessarily long time to complete.

The second method uses an undetermined number of variable time subincrements which are calculated throughout the creep analysis by the CSTEM code to satisfy certain user defined

constraints on stress increment, creep strain increment, and integration error over the time subincrement. The length of time in each subincrement is increased or decreased as necessary so that the maximum change over the time subincrement in each of these constrained quantities is less than a specified amount.

The constraint on stress increment makes the assumption that stress varies linearly with time. The size of the next time increment is calculated from the size of the just completed time increment using the ratio of the allowable change in effective stress to the maximum change in effective stress at an integration point which occurred over the time increment,  $\delta t_k$ . The next time subincrement,  $\delta t_{k+1}$ , is then calculated as

$$\delta t_{k+1} = \delta t_k \frac{(\Delta \sigma_{eff})_{allowable}}{(\Delta \sigma_{eff})_{max}} \quad (101)$$

The constraint on creep strain increment is similar to the stress constraint. It also makes the assumption of linear change with time. The size of the next time increment is determined from the previous time increment using the ratio of allowable change in effective creep strain to the maximum change in effective creep strain at an integration point which occurred over the time increment,  $\delta t_k$ . The next time subincrement,  $\delta t_{k+1}$ , is then calculated as

$$\delta t_{k+1} = \delta t_k \frac{(\Delta \epsilon_{eff}^c)_{allowable}}{(\Delta \epsilon_{eff}^c)_{max}} \quad (102)$$

The constraint on integration error assumes a second order error estimate, which is

$$error = \frac{\ddot{\epsilon}_{eff}^c}{2!} \delta t^2 \quad (103)$$

The second time derivative in creep strain is estimated as

$$\ddot{\epsilon}_{eff}^c = \frac{(\dot{\epsilon}_{eff}^c)_{t+\delta t} - (\dot{\epsilon}_{eff}^c)_t}{\delta t} \quad (104)$$

The maximum allowable integration error multiplied by the maximum allowable creep strain increment over the time subincrement gives the error tolerance in terms of creep strain increment,

so that the next time subincrement based on integration error is calculated as

$$\delta t_{k+1} = \sqrt{\left[ \frac{2! \delta t_k}{(\dot{\epsilon}_{eff}^c)_{t+\delta t} - (\dot{\epsilon}_{eff}^c)_t} \right] tol * (\epsilon_{eff}^c)_{allowable}} \quad (105)$$

The assumptions of linear change over time may be less accurate as the time increment increases. For this reason a limit on the increase in time subincrement size is imposed. This is a user specified limit, which defaults to a value of 1.5.

#### 3.1. 11. 4.2. Creep Formulation

The creep formulation implemented in CSTEM actually calculates the creep strain rate, found by taking the time derivative of the creep equation and using the average of creep strain rate at the beginning and end of the time subincrement to calculate the creep strain increment. This is called a 2nd order Adams – Moulton method.

The change in stress from one time increment to another is accounted for in the rate form of the creep equations. Changes in temperature are accounted for by interpolation of the creep constants to the temperature of the integration point. This changes the actual creep curve being used. The method used to shift from one creep curve to another is called the creep hardening formulation. Two choices of creep hardening are available in CSTEM: time hardening and strain hardening. Time hardening transitions from one creep curve to another by keeping the accumulated creep time constant. Since creep strain increments are calculated, the creep strain increment in the ensuing time subincrement will occur at the rate on the new creep curve at the point which corresponds to the accumulated creep time. Strain hardening transitions from one creep curve to another by keeping the accumulated creep strain constant. The creep rate increment in the ensuing time subincrement will occur at the rate on the new creep curve at the point which corresponds to the same accumulated effective creep strain. The secant method is used to determine the equivalent creep time on the new creep curve corresponding to the accumulated effective creep strain. Figure 15. illustrates these two hardening formulations.

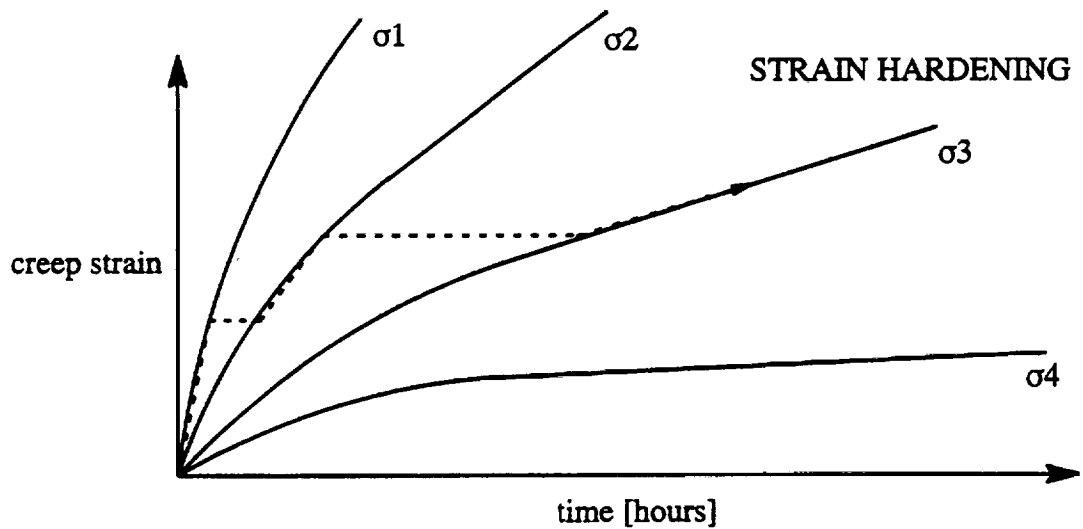
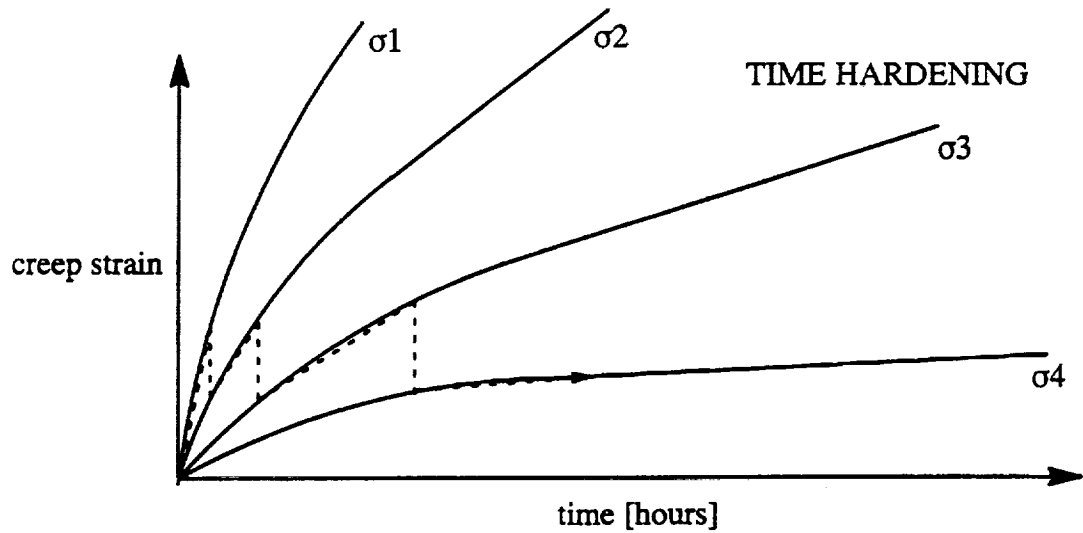


FIGURE 15. CREEP HARDENING FORMULATIONS

Convergence is based on effective stress values at each integration point. The difference in effective stress from one iteration to the next must be less than a user specified tolerance for convergence.

#### 3.1. 11. 4.3. Isotropic Material

The isotropic creep model is a 5 constant model involving two terms, one dominated by primary creep and the other dominated by secondary creep. This model can be written as

$$\epsilon_{eff}^c = K \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^n t^m + Q \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^r t \quad (106)$$

where  $\sigma_{norm}$  is a scalar stress normalization factor in psi,  $t$  is time in hours,  $\sigma_{eff}$  is the effective stress in psi at the integration point, and  $\epsilon_{eff}^c$  is the effective creep strain in inches/inch. The 5 creep constants, ( $K$ ,  $n$ ,  $m$ ,  $Q$ , and  $r$ ), are temperature and material dependent, input at up to ten different temperatures. The primary creep term is of the same form as the Bailey–Norton creep model. By specifying constants to remove the secondary creep term ( $Q=0$ ), the model becomes the Bailey–Norton creep model.

Expansion of the effective creep strain increment into its creep strain components is done using the same stress based Prandtl–Reuss flow rule used with isotropic plasticity.

#### 3.1. 11. 4.4. Orthotropic Material

The orthotropic creep model in CSTEM is the C.T. Sun orthotropic creep model. Similar to the implementation of the orthotropic plasticity model, the orthotropic creep model has the capability to analyze response to varying non–uniform temperature loading and cyclic loading of 3D layered finite element models containing multiple materials. The creep model utilizes the same potential function and effective stress definition as the plasticity model (equations (93) and (96)). Uniaxial creep tests of unidirectional aligned and off axis specimens are used to generate the material creep parameters. The creep law is defined as:

$$\epsilon_{eff}^c = C \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^n \left[ 1 - \exp \left[ - D \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^r t^s \right] \right] + B \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^m t \quad (107)$$

where  $\sigma_{norm}$  is a scalar stress normalization factor in psi,  $t$  is time in hours,  $\sigma_{eff}$  is the effective stress in psi at the integration point, and  $\epsilon_{eff}^c$  is the effective creep strain in inches/inch. The seven coefficients in the creep law, ( $C$ ,  $n$ ,  $D$ ,  $r$ ,  $s$ ,  $B$ , and  $m$ ), are input at up to ten different temperatures for each material. Calculations are performed relative to the coordinate system of the material. The initial stiffness method is used to avoid the expensive reformulation of stiffnesses. The user may select strain hardening or time hardening creep formulations.

The associated flow rule utilized for creep is a rate form of the same flow rule used in the orthotropic plasticity formulation:

$$\dot{\epsilon}_{ij}^c = \lambda \frac{\partial f}{\partial \sigma_{ij}} \quad \text{where} \quad \lambda = \frac{3}{2} \sigma_{eff} \dot{\epsilon}_{eff}^c \quad (108)$$

A more detailed description of this creep model can be found in "Modeling Creep in Thermoplastic Composites", Ilusup Chung, C.T. Sun, and I.Y. Chang, Journal of Composite Materials, Vol. 27, No. 10, 1993.

#### 3.1. 11. 4.5. CRHOOK User Creep

A method is available by which a user's custom creep model can be included for use in CSTEM. User creep requires the user to compile the user creep model with the CSTEM code. The user creep model receives information from CSTEM and returns information to CSTEM through a single routine called CRHOOK, which links the user creep model into the CSTEM iteration scheme. The CRHOOK routine is identical to the user plasticity routine CHOOK, except that it is accessed from the creep analysis iteration scheme rather than the plasticity analysis iteration scheme. The calling arguments of the CRHOOK routine contain vectors which are sized by user specification of certain input deck variables. These vectors contain some predefined quantities, such as stress, strain, temperature, time, etc., as well as user defined information about the current state of the material. The user updates and stores whatever internally generated user defined information is needed by the model in these vectors within the vector lengths defined by the input deck variables. The vectors are returned to CSTEM and stored on a random access file for each calculation location. Calculations are performed at each integration point in the finite element model. Figure 13. shows the CHOOK calling sequence and predefined vector locations, which are identical to the CRHOOK calling sequence and predefined vector locations. User data is stored after the predefined data locations. The UMATIO routine concept shown in Figure 14. can be utilized by the user creep model for access and storage of user material data.



### 3.2. HEAT TRANSFER ANALYSIS

#### 3.2. 1. Introduction

In recent years the finite element idealization has become a general approach for the stress analysis of complex structural systems. In order to minimize the preparation of data for a thermal stress problem it is desirable that the same finite element model be used in the stress and heat transfer analysis. For this reason considerable effort is currently being devoted to the development of compatible heat transfer and stress analysis programs.

In general, the stress and heat transfer analysis of solids are coupled. The coupling of structural mechanical loads, heat transfer, and electromagnetic are discussed elsewhere. Here, we are only talking about the heat transfer portion of the entire program. In heat transfer analysis nonlinearities may be due to temperature dependent material properties and, in particular, be caused by nonlinear boundary conditions.

The heat transfer analysis of graded composite structural problems idealized by finite elements is currently in the developmental and experimental stage. The procedures of the treatment of temperature dependent material properties and the nonlinear boundary conditions in graded composite structures are covered in the following text.

The purpose of this section is to explain development of the techniques that permit the practical analysis of complex three dimensional heat transfer problems of graded composite structures.

#### 3.2. 2. Finite Element Implementation

The three dimensional isoparametric solid elements are used so that the mesh size can be varied and bodies of arbitrary shape can be considered without difficulty. Material properties can be different for each element. In general, mixed boundary conditions can be handled directly. The equations which govern the response of the discrete system generally involve matrices which are symmetric and positive definite. Therefore, effective solution techniques can be employed for the solution of both the steady state and transient problems.

An 8, 16, and 20 node, three dimensional solid element is shown in Figure 16. The natural coordinates (r,s,t) of the eight corner nodes are  $(\pm 1, \pm 1, \pm 1)$  and of the 12 remaining nodes are  $(0, \pm 1, \pm 1)$ ,  $(\pm 1, 0, \pm 1)$ ,  $(\pm 1, \pm 1, 0)$ . The temperature within the element is defined in terms of the nodal temperatures by

$$\Theta(X, Y, Z, t) = \sum H_i(X, Y, Z, t) \Theta_i(t) \quad (109)$$

A detailed description of the shape functions can be found in Section 2.4.1. in the description of 3D elements. In the heat transfer analysis, the matrix [B] is not the same as in a structural analysis and is formed as

$$[B] = [J] \begin{bmatrix} \frac{\partial H_1}{\partial r} & \dots & \frac{\partial H_i}{\partial r} \\ \frac{\partial H_1}{\partial s} & \dots & \frac{\partial H_i}{\partial s} \\ \frac{\partial H_1}{\partial t} & \dots & \frac{\partial H_i}{\partial t} \end{bmatrix} \quad (110)$$

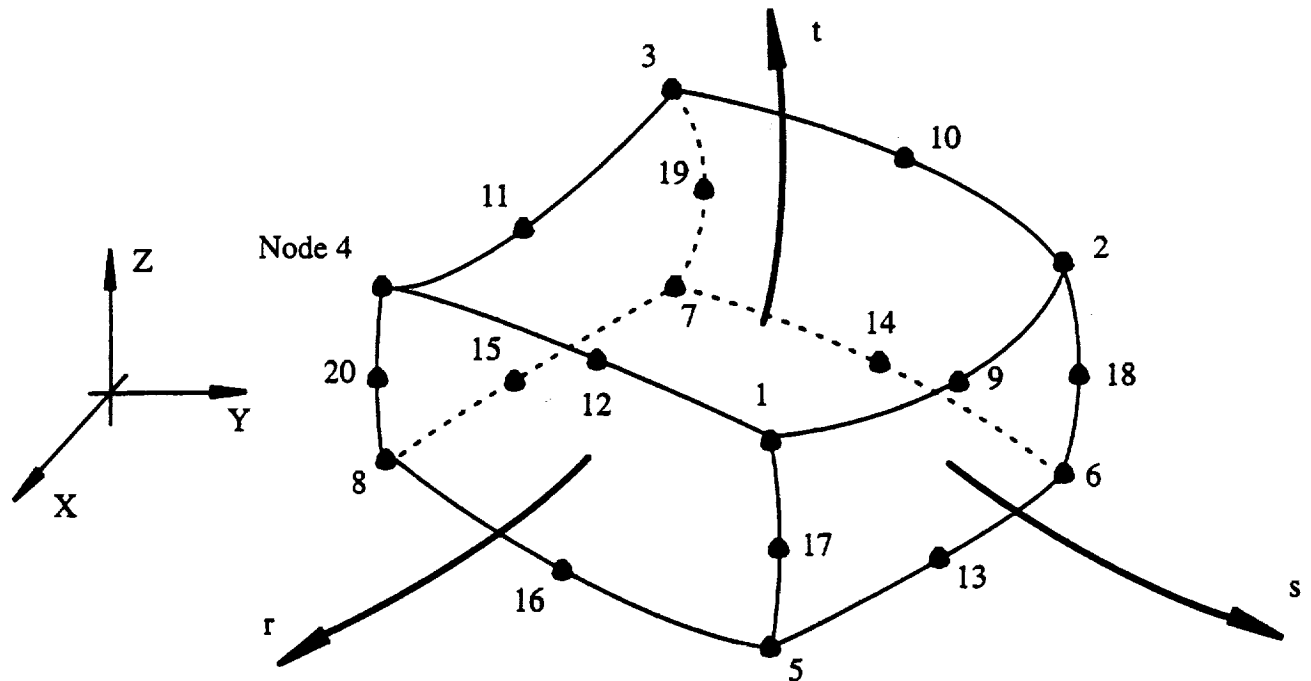


FIGURE 16. 3D ISOPARAMETRIC ELEMENT (8, 16, or 20 NODES)

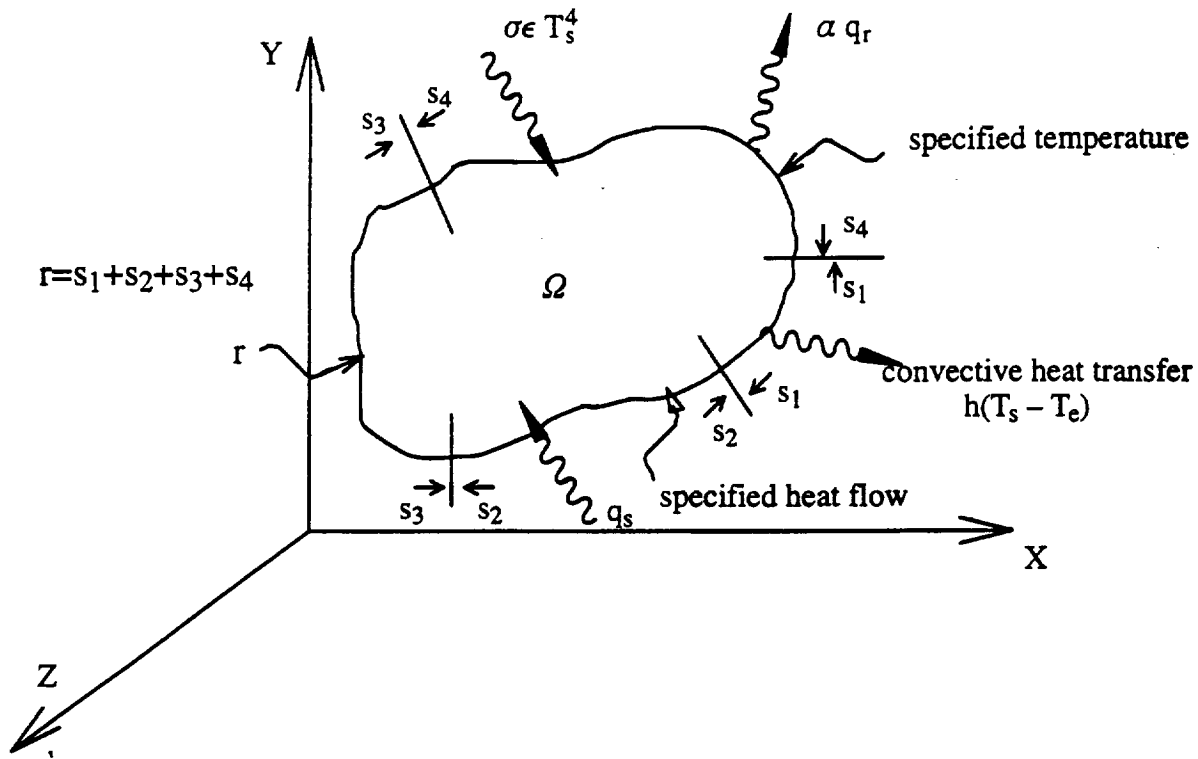


FIGURE 17. 3D SOLUTION DOMAIN FOR GENERAL HEAT CONDUCTION

### 3.2. 3. Equations for Heat Transfer Analysis

Consider the three dimensional heat transfer conditions as shown in Figure 17. In the analysis of heat transfer conditions, we assume that the material obeys Fourier's law of heat conduction:

$$q_x = -k_x \frac{\partial \theta}{\partial x} \quad q_y = -k_y \frac{\partial \theta}{\partial y} \quad q_z = -k_z \frac{\partial \theta}{\partial z} \quad (111)$$

where  $q_x$ ,  $q_y$ , and  $q_z$  are the heat flows conducted per unit area,  $\theta$  is the temperature of the body, and  $k_x$ ,  $k_y$ , and  $k_z$  are the thermal conductivities corresponding to the global axes X, Y, and Z.

Considering the heat flow equilibrium in the interior of the body we thus obtain

$$\frac{\partial}{\partial x} (k_x \frac{\partial \theta}{\partial x}) + \frac{\partial}{\partial y} (k_y \frac{\partial \theta}{\partial y}) + \frac{\partial}{\partial z} (k_z \frac{\partial \theta}{\partial z}) = -q_b \quad (112)$$

where  $q_b$  is the rate of heat generated per unit volume. On the surfaces of the body the following conditions must be satisfied:

$$\theta|_{s1} = \theta_e \quad (113)$$

$$k_n \frac{\partial \theta}{\partial n}|_{s2} = q_s \quad (114)$$

where  $\theta_e$  is the environmental temperature,  $k_n$  is the body thermal conductivity,  $n$  denotes the direction of the normal to the surface (outward) and  $q_s$  is the heat flow input to the surface of the body.

#### 3.2. 4. Boundary Conditions

1. Temperature Conditions – The temperature may be prescribed at specific points and surfaces of the body, denoted by  $s1$  in equation (113).
2. Heat Flow Conditions – The heat flow input may be prescribed at specific points and surfaces of the body. The heat flow boundary conditions are specified in equation (114).
3. Convection Boundary Conditions – Included in equation (114) are convection boundary conditions, where

$$q_s = h (\theta_e - \theta_s) \quad (115)$$

4. Radiation Boundary Conditions – Radiation boundary conditions are also specified in equation (114). Two different radiation conditions are considered; radiation from one surface to another surface and radiation from a gas to a surface. Surface to surface radiation is described using

$$q_r = \beta \epsilon_s f (\theta_r^4 - \theta_s^4) \quad (116)$$

while gas to surface radiation is described using

$$q_r = \beta \epsilon_s f (\epsilon_g \theta_r^4 - \alpha_g \theta_s^4) \quad (117)$$

where the variable  $\beta$  is the Stefan–Boltzman constant,  $\epsilon_s$  is the emissivity of the absorbing surface,  $\epsilon_g$  and  $\alpha_g$  are the emissivity and absorptivity of the radiating gas, and  $f$  is the geometric view factor.

In addition to these boundary conditions, the initial conditions must also be specified in a transient analysis.

### 3.2. 5. Solution Scheme

For the development of a finite element scheme either a Galerkin formulation operating on the differential equation of equilibrium or a variational formulation of the heat transfer problem can be employed. In the variational formulation a function  $\Pi$  is defined such that when invoking the stationarity of  $\Pi$ , the governing differential equations (1) to (3) are obtained.

$$\Pi = \int_V \frac{1}{2} \left[ k_x \left( \frac{\partial \theta}{\partial x} \right)^2 + k_y \left( \frac{\partial \theta}{\partial y} \right)^2 + k_z \left( \frac{\partial \theta}{\partial z} \right)^2 \right] dV - \int_V \theta q_b dV - \int_S \theta_s q_s dS - \sum_i \theta_i Q_i \quad (118)$$

where  $\theta_i$  are the concentrated heat flow inputs. Using the condition of stationarity of  $\Pi$  we obtain

$$\int \partial \theta'^T \mathbf{k} \theta' dv = \int \partial \theta q_b dv + \int \partial \theta_s q_s ds + \sum_i \partial \theta_i Q_i \quad (119)$$

where  $\theta'$  is the temperature gradient vector and  $\mathbf{k}$  is the conductivity matrix.

For a general solution scheme of both linear and nonlinear, steady state and transient problems we aim to develop incremental equilibrium equations. An incremental iteration scheme

for heat flow equilibrium is used, in which

$${}^{t+\Delta t}\theta^{(i)} = {}^{t+\Delta t}\theta^{(i-1)} + \Delta\theta^{(i)} \quad (120)$$

with the initial condition

$${}^{t+\Delta t}\theta^{(0)} = {}^t\theta \quad (121)$$

and  $\Delta\theta^{(i)}$  is the temperature increment. In the iteration procedure, the conductivity matrix,  $[K]$ , and the right hand side load vectors,  $\{Q\}^{(i)}$ , are updated at each iteration as necessary. The COLSOL technique is used to solve the matrix set of equations for the nodal temperature.

There are four generic heat transfer problems, which are discussed in the following:

#### I. Linear Steady State Conditions

The governing equation of linear steady state can be written as

$$(K^k + K^c) {}^{t+\Delta t}\theta = {}^{t+\Delta t}Q + {}^{t+\Delta t}Q_e \quad (122)$$

where  $K^k$  is the conductivity matrix

$$K^k = \sum_m \int_V B^T k_m B dV \quad (123)$$

and  $K^c$  is the convection matrix

$$K^c = \sum_m \int_S h H_s^T H_s dS \quad (124)$$

in which summations are over the  $m$  elements.

The nodal point heat flow input vector,  ${}^{t+\Delta t}Q$ , is defined as

$${}^{t+\Delta t}Q = {}^{t+\Delta t}Q_b + {}^{t+\Delta t}Q_s + {}^{t+\Delta t}Q_c \quad (125)$$

where

$${}^{t+\Delta t}\mathbf{Q}_b = \sum_m \int_V \mathbf{H}^T {}^{t+\Delta t}q_b dV \quad (126)$$

and

$${}^{t+\Delta t}\mathbf{Q}_s = \sum_m \int_S \mathbf{H}_s^T {}^{t+\Delta t}q_s dS \quad (127)$$

and  ${}^{t+\Delta t}\mathbf{Q}_c$  is a vector of concentrated nodal point heat flow input,  $q_b$  is the rate of heat generated in the element and  $q_s$  is the surface heat flow input.

The nodal point heat flow contribution due to the convection boundary conditions,

${}^{t+\Delta t}\mathbf{Q}_e$ , is found using

$${}^{t+\Delta t}\mathbf{Q}_e = \sum_m \int_S h \mathbf{H}_s^T \mathbf{H}_s {}^{t+\Delta t}\theta_e \quad (128)$$

where  ${}^{t+\Delta t}\theta_e$  are the given environmental temperature at the nodal points and  $h$  is the convection coefficient as mentioned previously.

## II. Nonlinear Steady State Conditions

The governing equation of nonlinear steady state heat transfer is

$$\left[ {}^t\mathbf{K}^k + {}^t\mathbf{K}^c + {}^t\mathbf{K}^r \right] \Delta\theta^{(i)} = {}^{t+\Delta t}\mathbf{Q} + {}^{t+\Delta t}\mathbf{Q}_c^{(i-1)} + {}^{t+\Delta t}\mathbf{Q}_r^{(i-1)} - {}^{t+\Delta t}\mathbf{Q}_k^{(i-1)} \quad (129)$$

where the conductivity matrix is defined as

$${}^t\mathbf{K}^k = \sum_m \int_V \mathbf{B}^T {}^t\mathbf{k}_m \mathbf{B} dV \quad (130)$$

the convection matrix is defined as

$${}^t\mathbf{K}^c = \sum_m \int_S {}^t h \mathbf{H}_s^T \mathbf{H}_s dS \quad (131)$$

and the radiation matrix is defined as

$${}^t\mathbf{K}^r = \sum_m \int_S {}^t \kappa \mathbf{H}_s^T \mathbf{H}_s dS \quad (132)$$

The nodal point heat flow vector is the same as in the linear steady state case, namely

$${}^{t+\Delta t}\mathbf{Q} = {}^{t+\Delta t}\mathbf{Q}_b + {}^{t+\Delta t}\mathbf{Q}_s + {}^{t+\Delta t}\mathbf{Q}_c \quad (133)$$

The convection load vector is defined as

$${}^{t+\Delta t}\mathbf{Q}_c^{(i-1)} = \sum_m \int_S {}^{t+\Delta t} h^{(i-1)} [\mathbf{H}_s ({}^{t+\Delta t}\theta_e - {}^{t+\Delta t}\theta^{(i-1)})] dS \quad (134)$$

The radiation load vector is defined as

$${}^{t+\Delta t}\mathbf{Q}_r^{(i-1)} = \sum_m \int_S {}^{t+\Delta t} \kappa^{(i-1)} \mathbf{H}_s^T [\mathbf{H}_s ({}^{t+\Delta t}\theta_r - {}^{t+\Delta t}\theta^{(i-1)})] dS \quad (135)$$

for surface to surface radiation, where  $\kappa$  is the radiation coefficient such that

$$\kappa = \beta \epsilon_s f (\theta_r^2 + \theta_s^2) (\theta_r + \theta_s) \quad (136)$$

and  $\beta$  is the Stefan – Boltzmann constant,  $\epsilon_s$  is the emissivity of the surface,  $f$  is the viewfactor,

$\theta_s$  is the surface temperature, and  $\theta_r$  is the temperature of the radiating surface.

A slightly different formulation is used for radiation from a gas to a surface. In this case the radiation load vector is calculated as

$${}^{t+\Delta t}\mathbf{Q}_r^{(i-1)} = \sum_m \int_S {}^{t+\Delta t} K^{(i-1)} \mathbf{H}_s^T [\mathbf{H}_s ({}^4\sqrt{\epsilon_g} {}^{t+\Delta t}\theta_r - {}^4\sqrt{\alpha_g} {}^{t+\Delta t}\theta^{(i-1)})] dS \quad (137)$$



where  $K$  is the radiation coefficient such that

$$K = \beta \epsilon_s f (\sqrt{\epsilon_g} \theta_r^2 + \sqrt{\alpha_g} \theta_s^2) (4\sqrt{\epsilon_g} \theta_r + 4\sqrt{\alpha_g} \theta_s) \quad (138)$$

and  $\beta$  is the Stefan – Boltzmann constant,  $\epsilon_s$  is the emissivity of the surface,  $f$  is the viewfactor,  $\theta_s$  is the surface temperature,  $\theta_r$  is the temperature of the radiating gas,  $\epsilon_g$  is the emissivity of the radiating gas, and  $\alpha_g$  is the absorptivity of the radiating gas.

The internal thermal load vector is defined as

$${}^{t+\Delta t} \mathbf{Q}_k^{(i-1)} = \sum_m \int_V \mathbf{B}^T [{}^{t+\Delta t} \mathbf{k}_m^{(i-1)} \mathbf{B} {}^{t+\Delta t} \theta^{(i-1)}] dV \quad (139)$$

The thermal conductivity and radiation emissivity are temperature dependent and the convection coefficients are both temperature and time dependent. Because of this an iterative procedure must be used to solve these equations. A Newton–Raphson method is used to iterate the following equation to convergence:

$${}^{t+\Delta t} \theta^{(i)} = {}^{t+\Delta t} \theta^{(i-1)} + {}^{t+\Delta t} \Delta \theta^{(i)} \quad (140)$$

Convergence is evaluated by both of the following criteria:

$$\begin{aligned} |\theta^{(i)} - \theta^{(i-1)}| &\leq \delta_1 \\ \left| \frac{\theta^{(i)} - \theta^{(i-1)}}{\theta^{(i-1)}} \right| &\leq \delta_2 \end{aligned} \quad (141)$$

The conductivities, convection coefficients, and radiation coefficients are input data. They are set up in data tables and stored on file for use during the iterative procedure.

### III. Transient Analyses

In transient heat transfer analysis the heat capacity effects must be included in the analysis as part of the rate of heat generated. If the Euler backward implicit time integration is employed, the heat flow equilibrium equations used are obtained directly from the equations of steady state

conditions:

$$\dot{\Theta}_m(X, Y, Z) = H_m(X, Y, Z) \dot{\Theta}(t) \quad (142)$$

then

$${}^{t+\Delta t}Q_b = \sum_m \int_V H^T ({}^{t+\Delta t}q_b - {}^{t+\Delta t}c_m H {}^{t+\Delta t}\Theta) dV \quad (143)$$

where  ${}^{t+\Delta t}q_b$  no longer includes the rate at which heat is stored within the material. Thus, the transient conditions are, in linear analysis:

$$C {}^{t+\Delta t}\dot{\Theta} + (K^k + K^c) {}^{t+\Delta t}\Theta = {}^{t+\Delta t}Q + {}^{t+\Delta t}Q_e \quad (144)$$

and in nonlinear analysis:

$${}^{t+\Delta t}C^{(i)} {}^{t+\Delta t}\dot{\Theta}^{(i)} + ({}^tK^k + {}^tK^c + {}^tK^r) \Delta\Theta^{(i)} = {}^{t+\Delta t}Q + {}^{t+\Delta t}Q_c^{(i-1)} + {}^{t+\Delta t}Q_r^{(i-1)} - {}^{t+\Delta t}Q_k^{(i-1)} \quad (145)$$

where  $C$  and  ${}^{t+\Delta t}C^{(i)}$  are the heat capacity matrices defined as

$$C = \sum_m \int_V H^T c_m H dV \quad (146)$$

$${}^{t+\Delta t}C^{(i)} = \sum_m \int_V H^T {}^{t+\Delta t}c_m^{(i)} H dV \quad (147)$$

### 3.2. 6. Graded Composite Structures

The structure is built up from a series of layers of different materials, such that the material properties are discontinuous functions of the thickness variable,  $\xi$ . This requires that an appropriate integration through the thickness must be performed. For the CSTEM finite element code, we use a layered approach wherein a midpoint rule integration scheme is adopted for each layer.

Layers are numbered sequentially, starting at the bottom surface of the element. The bottom surface is the surface at the negative end of the through thickness structural (rst) axis. Each layer contains integration points on its midsurface. The thermal matrices and load vectors are computed at these integration points and integrated into the elemental matrices and load vectors. The thermal gradient distribution through the thickness of the element is controlled by the shape function of the element. Layers of different thicknesses can be used, as well as different numbers of layers per element. However, consideration must be given to the capability to accurately integrate the varying thermal properties of the layers. In general, layers of relatively equal thickness will provide a more accurate integration than a mixture of relatively thick and thin layers within an element.

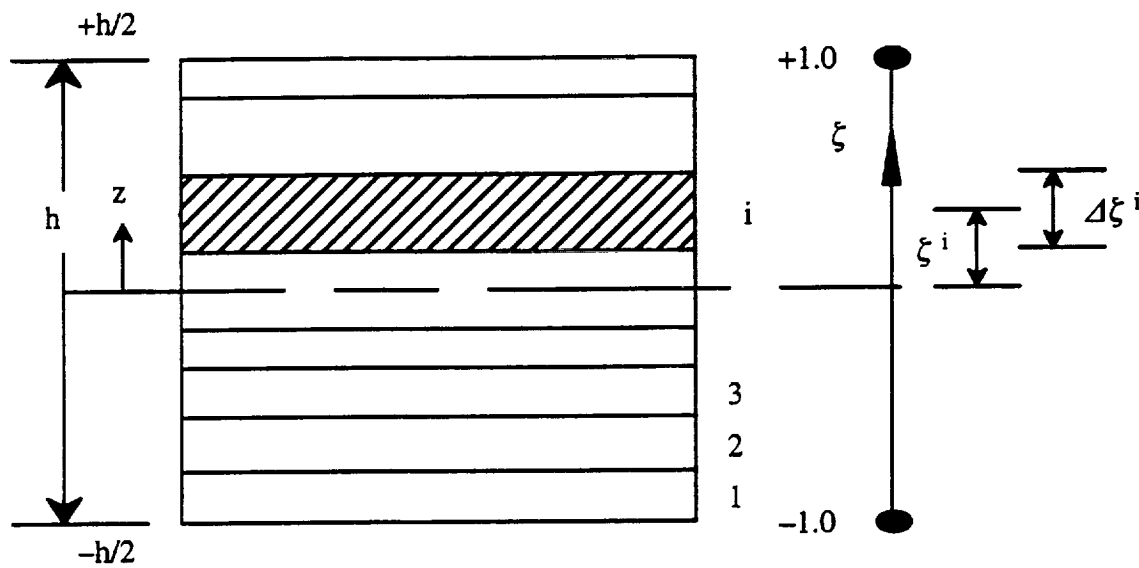


FIGURE 18. LAYERED MODEL

The matrices for the heat transfer equations are obtained by the integration of each layer of the element in the same way as shown in Section 2.4.3. For orthotropic materials, transformation of the conductivity matrix must be performed to orient the matrix to the global directions.

The elemental thermal conductivity matrix can be written as follows:

$$[K] = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \quad (148)$$

The transformation of this matrix can be written as

$$[K]_G = [T]^T [K] [T] \quad (149)$$

where  $[T]$  is the coordinate transformation matrix as described in Section 2.3.1.

### 3.3. ELECTROMAGNETIC ANALYSIS

Three different methods for calculating the electromagnetic response of a structure are available in CSTEM. All three of the methods are applicable for a particular wave impinging on a point of the structure. Calculations are done for one given frequency, orientation, and path of an electromagnetic wave at a time. Multiple frequencies and/or different paths travelled through the structure would be handled by separate calculations for the different parameters.

The orientation of the wave is input in the same manner as skew boundary conditions where a wave coordinate system is associated with the electromagnetic wave. This wave coordinate system is defined such that the direction of propagation is along the positive z axis and polarization is measured from the positive x axis as shown in Figure 5. Polarization of the wave is defined only from 0 to 180 degrees. The limit on the number of different wave specifications per load case is set by the parameter MAXW.

The impingement locations can either be input specifically or an exposure analysis can be done so that an analysis for all exposed faces will be done. Specific input of an impingement face is not checked to determine whether or not it is a surface face. The limit on the number of different specifically input impingement locations per load case is set by the parameter MAXP. For an exposure analysis there is no limit on the number of faces, but there is a cutoff exposure value (IEXCUT) below which an analysis for the face will not be done. There is also a check on the impingement angle so that the analysis of a wave with propagation direction within 1 degree of being parallel to an impingement face will not be calculated. When using layered elements, the impingement faces cannot be faces perpendicular to the layering.

The stacking sequence encountered by the propagating electromagnetic wave is determined by starting at the centroid of a given exposed element face and progressing through the cross section element by element. The elements through the cross section are determined by finding the element which shares the nodes on the face opposite to the surface face and continuing until another surface face is reached, or until a material specified as a conductor is encountered.

### 3.3. 1. Data Bank

Electromagnetic material properties are obtained through a data bank. The maximum number of materials that can be read in any given analysis is set by the parameter NMAT. Different formats for the data bank are available, depending on the method being used. The first two methods of calculating the electromagnetic response require a data bank containing the complex permittivity and permeability of the material as a function of temperature and frequency. In addition, an option of specifying the permittivity and permeability using the dielectric constant and loss tangent is available. The table look up method requires a data bank listing the fraction of the energy a material attenuates as a function of temperature, frequency, and polarization angle. The type being used is determined internally by counting the number of inputs on a frequency line, which means that the different types can be mixed.

The material is specified by a 1 to 8 character identifier. The number of temperatures, frequencies, and polarization angles that can be stored internally from the data bank is limited. These limits are set by the parameters NTEMP, NFRQ, and NANG. A more detailed description of the data bank can be found with the absorption input descriptions.

### 3.3. 2. Exposure Analysis

The impingement points used in an electromagnetic analysis can be input specifically by identifying the surface element and face. This is subject to a limit set by the parameter MAXP on the number of points specified. It is also possible to determine all faces exposed to the incoming electromagnetic wave and perform an analysis on each one that has above a certain cutoff percentage exposed. This percentage is set internally by the variable IEXCUT.

The exposure analysis is done by considering the faces of an element where the faces are determined by the corner nodes only. The global coordinates of the corner nodes are rotated to a local system so that the local Z axis lies parallel to the direction of propagation of the incoming electromagnetic wave. The face is split into two triangles and an area coordinate system is set up for each of the triangles. The exposed amount of each side of these triangles is determined by comparing with all other faces and determining if the other faces cover it. The exposed amount of

the face is assumed to be the same as the amount of the boundary of the face which is visible from the viewpoint of the incoming wave.

The frontal area exposed to the wave can be determined by summing over the faces. The frontal area of each face is found as the area projection of the face on the local X–Y coordinate system plane (which is perpendicular to the wave propagation direction). This area is then modified according to the amount of the face visible to the incoming waves, and summed into the total exposed frontal area.

### 3.3. 3. Wave Matrix Method

The computer program WAVES is a computer program for determining the reflection and transmission properties of multilayer plane impedance boundaries which has been included in CSTEM. The manual for this program is included in the appendix and contains a detailed description of the calculations performed in the program. In the CSTEM installation, the stacking sequence details, material properties, and specification of the impinging wave are determined from the finite element model and CSTEM electromagnetic input and transferred into the program.

### 3.3. 4. Optics Method

A method for calculation of electromagnetic response that determines reflected amounts of electromagnetic energy as well as calculating the attenuation of the transmitted amount can be derived using equations similar to optics. Figure 5. shows an electromagnetic wave propagating in the +Z direction of the coordinate system attached to the wave. The electric field vector points in the +X direction and the magnetic field vector points in the +Y direction for a polarization of zero degrees.

The wave equations which apply to these vectors can be obtained using Maxwell's equations, and can be written as:

$$\nabla^2 \mathbf{H} = \gamma^2 \mathbf{H} \quad \text{and} \quad \nabla^2 \mathbf{E} = \gamma^2 \mathbf{E} \quad (150)$$

where the propagation constant,  $\gamma$ , can be written as:

$$\gamma = \alpha + j\beta \quad (151)$$

Here,  $\alpha$  is the attenuation factor and is found as:

$$\alpha = \omega \sqrt{\frac{\mu\epsilon}{2} \left( \sqrt{1 + \left(\frac{\sigma}{\omega\epsilon}\right)^2} - 1 \right)} \quad (152)$$

and  $\beta$  is the phase shift constant, written as:

$$\beta = \omega \sqrt{\frac{\mu\epsilon}{2} \left( \sqrt{1 + \left(\frac{\sigma}{\omega\epsilon}\right)^2} + 1 \right)} \quad (153)$$

In the previous equations,  $\omega$  is the frequency of the electromagnetic wave in radians/second,  $\mu$  is the permeability of the material in henries/inch,  $\epsilon$  is the permittivity of the material in farads/inch, and  $\sigma$  is the conductivity of the material in siemens/inch.

The index of refraction of a material is the ratio of the wavelength or phase velocity in free space to that in the material. Since the wavelength can be written in terms of the phase shift constant as:

$$\gamma = \frac{2\pi}{\beta} \quad (154)$$

the index of refraction can be written in terms of the phase shift constant as:

$$n = \frac{\beta}{\beta_0} \quad (155)$$

where  $\beta$  is the phase shift factor for the dielectric material and  $\beta_0$  is the value of the phase shift constant for free space found as:

$$\beta_0 = \delta \sqrt{\mu_0 \epsilon_0} \quad (156)$$



The calculation of the reflected and transmitted electromagnetic energies is done using Snell's law of refraction and the Fresnel formulas. Snell's law can be stated as:

$$n_1 \sin \theta = n_2 \sin \phi \quad (157)$$

where  $n_1$  and  $n_2$  are the index of refraction of material 1 and 2,  $\theta$  is the angle of incidence of an electromagnetic wave propagating from material 1 to material 2, and  $\phi$  is the angle of refraction of the transmitted portion of the electromagnetic wave. Figure 19. shows the geometry associated with an impinging electromagnetic wave.

The Fresnel formulas assume an impinging wave with components of the electric vector that are in and out of the plane of incidence defined in Figure 19. This impinging wave is written as:

$$\mathbf{A}_1 = A_{p1} (\cos \theta \mathbf{i} - \sin \theta \mathbf{k}) + A_{s1} \mathbf{j} \quad (158)$$

where  $A_{p1}$  is the in plane amplitude and  $A_{s1}$  is the out of plane amplitude, and  $\theta$  is the angle of incidence on the interface between materials 1 and 2. The reflected portion of this wave can then be written:

$$\mathbf{A}_2 = A_{p2} (\cos \theta \mathbf{i} + \sin \theta \mathbf{k}) + A_{s2} \mathbf{j} \quad (159)$$

where

$$A_{p2} = -A_{p1} \frac{\tan(\theta - \phi)}{\tan(\theta + \phi)} \quad (160)$$

$$A_{s2} = -A_{s1} \frac{\sin(\theta - \phi)}{\sin(\theta + \phi)} \quad (161)$$

The refracted wave can be similarly written as:

$$\mathbf{A}_3 = A_{p3} (\cos \phi \mathbf{i} + \sin \phi \mathbf{k}) + A_{s3} \mathbf{j} \quad (162)$$

where

$$A_{p3} = A_{p1} \frac{2 \sin \phi \cos \theta}{\sin(\theta + \phi) \cos(\theta - \phi)} \quad (163)$$

$$A_{s3} = A_{s1} \frac{2 \sin \phi \cos \theta}{\sin(\theta + \phi)} \quad (164)$$

For both the reflected and refracted waves,  $\theta$  is the angle of incidence and  $\phi$  is the angle of refraction obtained using Snell's law.

For normal incidence these equations can be written much more simply using the index of refraction,  $n$ . For the reflected wave:

$$A_2 = -A_1 \frac{n - 1}{n + 1} \quad (165)$$

and for the refracted wave:

$$A_3 = -A_1 \frac{2}{n + 1} \quad (166)$$

The attenuation (or absorption) of a transmitted wave can be found using the term,  $e^{-\alpha x}$ , where  $x$  is the distance travelled in the material with an attenuation factor of  $\alpha$ .

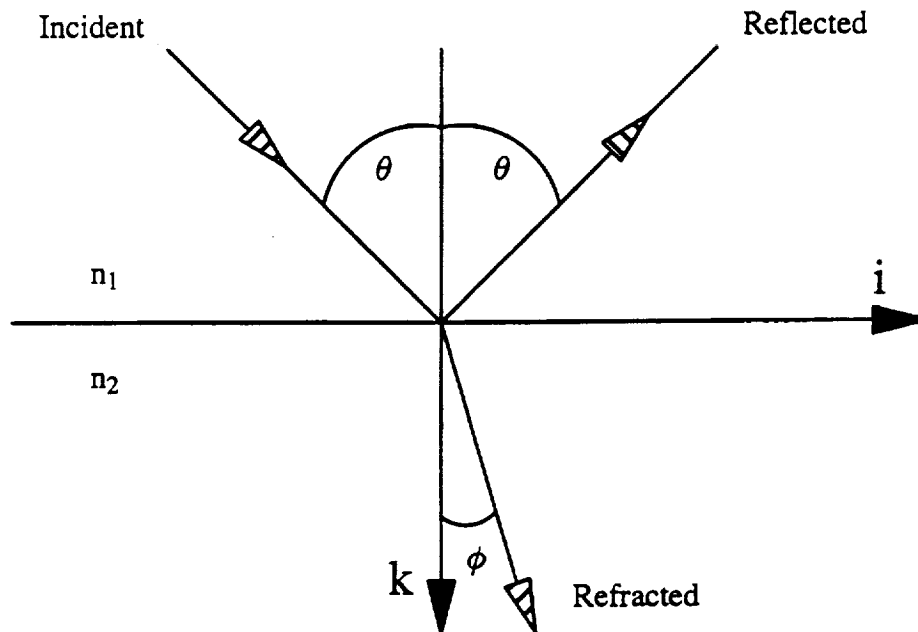


FIGURE 19. ELECTROMAGNETIC WAVE AT MATERIAL INTERFACE

### 3.3. 5. Table Look Up Method

This method is based on a data bank of absorptivity values for given material types specified at discrete values of temperature, frequency, and polarization angle. The information for a specific material or materials is read from the data bank file into tables which are used by the program. The absorptivity is linearly interpolated from these discrete values to the local values of temperature, frequency, and polarization angle. These local values are determined for each layer encountered in a cross section. The propagating wave is modified as determined by the values interpolated from the tables and these new values are then used with the next layer encountered. The fraction of energy loss of the wave as determined from the tables is applied only to the normal component of the impinging wave, while the horizontal component is assumed to be unaffected.

The impingement angle of the wave on a layer is calculated from the dot product of the inward normal of the layer midsurface and the propagation (+z) axis of the wave coordinate system. The impingement angle must be between 0 and 90 degrees in absolute value. The temperature is calculated using the element shape functions.

The polarization angle is calculated using the dot product between the projection of the wave polarization on the layer midsurface and the material principle direction. The polarization angle is calculated as being between 0 and 90 degrees.

The absorptivity values in the data bank are essentially percentage absorption values. The strength of the electromagnetic wave emerging on the other side of the layer is calculated as

$$W_{\text{out}} = W_{\text{in}} * (1.0 - \text{absorptivity}) \quad (167)$$

where  $W_{\text{in}}$  is the magnitude of the normal component of the impinging wave.

### 3.4. ACOUSTICS ANALYSIS

The sound power produced by a vibrating structure can be calculated in CSTEM. The surface of the structure is found automatically from the element connectivities. In addition, a portion of the surface can be eliminated from a sound power calculation by masking it out. This masking is useful if only a portion of the surface is considered to be radiating sound.

The natural frequencies and mode shapes of the structure are first found. Idealizing the element faces as rectangular flat plates, the radiation efficiencies and sound power of each mode for a range of point load forcing frequencies are determined based on the mode shapes and geometry of the structure. Once the radiation efficiencies are known, the sound power produced by a forced vibration of the structure can then be found by a modal summation of the contribution of each mode.

#### 3.4. 1. Theoretical Approach

The sound power calculated is based on the equation developed by Lord Rayleigh. This equation expresses the pressure field generated in a uniform free field by an extended plane surface. This integral equation may be written as

$$p(\mathbf{r}, t) = \frac{j\omega p_0}{2\pi} e^{j\omega t} \int_S \frac{\tilde{v}_n(\mathbf{r}_s) e^{-jkR}}{R} dS \quad (168)$$

where  $\mathbf{r}$  is the position vector of the observation point,  $\mathbf{r}_s$  is the position vector of an elemental surface  $dS$  having normal velocity  $\tilde{v}_n(\mathbf{r}_s)$ , and  $R$  is the distance between the observation point and the elemental surface.

In the far field, where  $R$  is much greater than the than the source size, an analytical solution to the previous integral equation has been produced by C. E. Wallace as described in the reference quoted later in this section. The source size in this case is defined as the largest length of an element face radiating sound. Figure 20. illustrates the situation considered in the acoustic

problem, in which the radiation from a modal vibration distribution at any arbitrary frequency is to be evaluated.

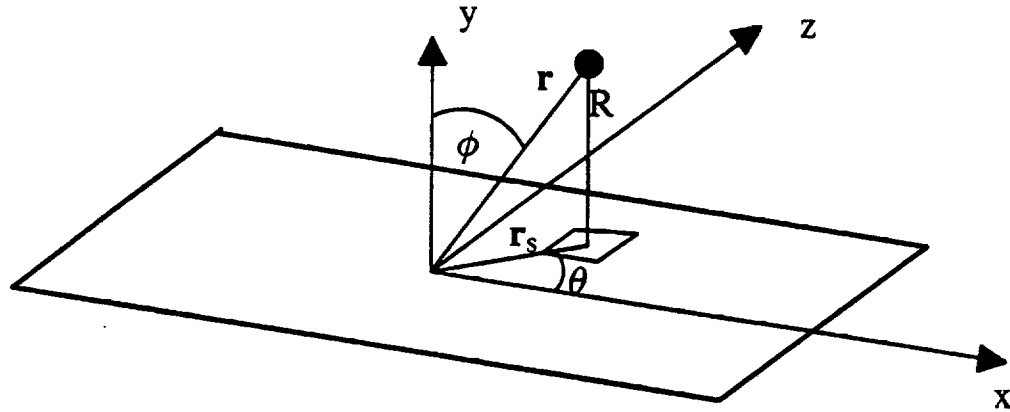


FIGURE 20. ILLUSTRATION OF ACOUSTIC PROB-

A differential area,  $dS$ , of a surface vibrating at a given frequency in an infinite baffle produces a differential pressure,  $dp$ , given by

$$dp = \frac{j\rho_0 c k}{2\pi R} [U dS] e^{-jkR} \quad (169)$$

where  $\omega$  is the frequency of vibration,  $\rho_0$  is the density of air ( $1.21 \times 10^{-7}$  lb<sub>f</sub>/in<sup>3</sup>),  $c$  is the speed of sound in air (13045. in/sec),  $k = \frac{\omega}{c}$  is the wave number,  $U$  is the peak surface velocity of the differential area,  $dS$  is the differential surface area,  $R$  is the distance from the differential area to the point in the free field at which the differential pressure exists. The total pressure at any point in the free field can be found by integrating  $dp$  over the vibrating surface:

$$p = \frac{j\rho_0 c k}{2\pi} \int_S \frac{e^{-jkR}}{R} [U dS] \quad (170)$$

At locations in the far field ( $kR \gg 1$ ), the sound power,  $W$ , for a surface freely vibrating at an arbitrary frequency  $\omega$  is

$$W = \int_A \frac{|P|^2}{2\rho_0 c} dA \quad (171)$$

where  $dA$  is a differential area of an imaginary surface enclosing the vibrating surface. In the CSTEM code this surface is a 10000 inch radius hemisphere. Note that the term being integrated in the above equation is known as the acoustic intensity, which is a measure of the amount of energy falling on a unit area normal to the direction of propagation.

A measure of the velocity of vibration is the space average value of the time average normal vibration velocity,  $\langle v_n^2 \rangle$ , calculated as

$$\langle v_n^2 \rangle = \frac{1}{2S} \int_S U^2 dS \quad (172)$$

Using the free vibration sound power and temporal and spatial average mean square velocity, the radiation efficiency at the particular frequency can be calculated as

$$\sigma = \frac{W}{\rho_0 c S \langle v_n^2 \rangle} \quad (173)$$

In most cases a structure is excited in such a way that certain vibratory modes are excited more than others. The CSTEM code uses a point load to preferentially excite certain modes of vibration. In this case, the mean square velocity in the vicinity of the exciting load generally exceeds the spatial average value used in calculating the radiation efficiency. The temporal and

spatial average mean square velocity for a point load excitation is

$$\langle v_n^2 \rangle = \frac{|\tilde{F}|^2}{2S} \sum_n \frac{\psi_n^2(\mathbf{r}_0) A_n}{|\tilde{Z}_n|^2} \quad (174)$$

In the above equation,  $\psi_n$  is the normalized modal velocity distribution (mode shape) of mode  $n$  and  $\mathbf{r}_0$  is the position vector to the point where the load is applied. The term  $A_n = \int \psi_n^2(\mathbf{r}) dS$

where  $\mathbf{r}$  is the position vector to an arbitrary point on the structure, and  $|\tilde{Z}_n|^2$  is the norm of the modal impedance. The modal impedance is given by

$$\tilde{Z}_n = jm_n(\omega - \frac{\omega_n^2}{\omega}) + \eta_n \frac{\omega_n^2}{\omega} \quad (175)$$

where  $\omega_n$  is the free vibration frequency for mode  $n$ ,  $\omega$  is the frequency of the point load,  $m_n$  is the generalized modal mass, which can be found as

$$m_n = \int_S m(\mathbf{r}) \psi_n^2(\mathbf{r}) dS \quad (176)$$

and  $\eta_n$  is the modal loss factor, a damping factor for mode  $n$ . Options for estimating modal damping include experimental results, assuming a constant value for all frequencies, or estimation from similar structures. The loss factor usually has values in the range  $5 \times 10^{-4}$  to  $5 \times 10^{-2}$  and for most structures the loss factor tends to decrease with frequency, roughly proportional to  $\omega^{-1/2}$

The sound power for a point load excited structure is then calculated as

$$\bar{P} = \sigma \rho_0 c S \langle v_n^2 \rangle \quad (177)$$

### 3.4. 2. Finite Element Implementation

An acoustics analysis requires an eigensolution of the free vibration problem for the structure. The number of free vibration modes  $\omega_n$  included in the analysis is indicated on the structural input deck. The acoustic input deck indicates loss factors for each free vibration



frequency considered. The force amplitude and direction, and the maximum forcing frequency, minimum forcing frequency, and the total number of frequencies to be logarithmically distributed between the max and min frequency are also input to preferentially excite certain modes of interest. An optional masking point can also be indicated to exclude certain portions of the structure from inclusion in the analysis.

Once the free vibration frequencies and mode shapes have been calculated, the free surfaces of the structure are determined. Any surfaces which can see the masking point (if one was input) are eliminated from the acoustic radiation calculations as well. The far field pressure generated by the structure is then calculated by integrating over each individual element surface face using a fourth order Gauss integration.

The far field observation points at which the pressure values are to be calculated are located on a hemisphere based on each element surface face. Using the coordinate system of Figure 20, the hemisphere is described by a radius,  $r = 10000$  inches, with the in plane angle,  $0 \leq \theta \leq 2\pi$ , and the out of plane angle,  $0 \leq \phi \leq \pi$ . These angles are subdivided into 31 increments each, creating 900 far field area segments over which the sound power is to be integrated. After the pressure field has been calculated, the radiation efficiency at each free vibration frequency is calculated by integrating over the hemisphere using Simpson's rule.

Finally, the acoustic impedance is calculated, and using this along with previously calculated quantities the sound power generated by each free vibration mode for the various forcing frequencies is obtained. The total sound power generated for each forcing frequency is obtained by summing the contribution of each of the free vibration modes considered.

### 3.4. 3. Acoustics References

Hayes, P.E., "Determining Vibration, Radiation Efficiency, and Noise Characteristics of Structural Designs Using Analytical Techniques", Proceedings – Society of Automotive Engineers p-106, SAE, Warrendale, PA, pp 263–270, 1982

Wallace, C.E., "Radiation Resistance of a Rectangular Panel", The Journal of the Acoustical Society of America, Vol. 51, No. 3 (Part 2), pp 946–952, 1972

Fahy, Frank, Sound and Structural Vibration Radiation, Transmission and Response, Academic Press, London, 1985

### 3.5. TAILORING METHODOLOGY

There are several numerical algorithms for solving the non-linear constrained optimization problem. Most of these algorithms assume that the design variables are continuous and that the objective and constraint functions are continuous with continuous first derivatives with respect to the design variables. Mathematically, the constrained optimization problem is stated as follows:

Find the set of design variables,  $\mathbf{X}$ , that will:

$$\text{Minimize or Maximize } F(\mathbf{X}) \quad (178)$$

Subject to:

$$g_j(\mathbf{X}) \leq 0 \quad j = 1, m \quad (179)$$

$$h_k(\mathbf{X}) = 0 \quad k = 1, l \quad (180)$$

$$x_i^l \leq x_i \leq x_i^u \quad i = 1, n \quad (181)$$

where  $F(\mathbf{X})$  is the objective function,  $g_j(\mathbf{X})$  and  $h_k(\mathbf{X})$  are inequality and equality constraints that define the criteria that must be satisfied for the design to be feasible.  $X^l$  and  $X^u$  are the lower and upper bounds on the design variables that define the region of search for the optimum. For example, the lower bound on the layer thickness may be set to a small positive number to prevent creation of a meaningless design. The design variables contained in  $\mathbf{X}$  may include layer thickness, layer orientation, fiber-volume ratios, etc. An example of an inequality constraint may be the strength constraint for the  $i^{\text{th}}$  layer and  $k^{\text{th}}$  loading condition and is given as follows:

$$g_j = U_{ki} - 1 \leq 0 \quad (182)$$

The typical optimization process proceeds by finding a search direction,  $\mathbf{S}$ , and then performing a one-dimensional search to find the point of maximum improvement by:

$$\mathbf{X}^q = \mathbf{X}^{q-1} + \alpha^* \mathbf{S}^q \quad (183)$$

where "q" is the iteration number and  $\alpha^*$  is the scalar multiplier on  $\mathbf{S}^q$  that defines how far we can move in this direction. In equation (183),  $\alpha^* \mathbf{S}^q$  is a perturbation on the design vector,  $\mathbf{X}^{q-1}$ . The

key is that optimization methods provide the tools to change all design variables simultaneously in search of the optimum.

In the modified method of feasible directions, the usable feasible search direction,  $S$ , is computed by solving the following subproblems:

- i) In the case when there are no active or violated constraints, a Fletcher–Reeves conjugate direction algorithm is used to find the search direction as:

$$S^q = -\nabla F(X^{q-1}) + \beta S^{q-1}$$

where

$$\beta = \frac{|\nabla F(X^{q-1})|^2}{|\nabla F(X^{q-2})|^2} \quad (184)$$

- ii) In the case when there are violated constraints, the direction finding problem is posed as:

$$\begin{aligned} \text{Minimize} \quad & \nabla F(X^{q-1}) \cdot S^q - \phi W \\ \text{Subject to} \quad & \nabla F(X^{q-1}) \cdot S^q + \theta_j W \leq 0 \quad j \in J \\ & S^q \cdot S^q + W^2 \leq 1 \end{aligned} \quad (185)$$

where  $q_j$  is called the "push off" factor since this determines the distance from the violated constraint and  $f$  is initially a small positive number that is gradually increased in order to bring the design to a feasible region. 'J' denotes the set of critical constraints at a design point  $X$ .

The feasible directions algorithm is based on the fact that the satisfaction of the Kuhn–Tucker conditions (finding a feasible stationary point of the Lagrangian function) is equivalent to finding at least a relative minimum of the original problem defined by equations (178) – (181). More details of the modified method of feasible directions can be obtained from the book by G.N. Vanderplaats, Numerical Optimization Techniques for Engineering Design: with Applications, McGraw Hill, New York, 1984.

## SECTION 4. DESCRIPTION OF INPUT

The CSTEM code obtains input from a number of input decks (files) which are fairly rigid in structure. Input variables are expected in a particular order, are generally numerical, and often involve reading a control variable (a flag) which tells the code how many lines of a particular type of input to read in the deck. Options are activated by the value of a variable or by the sign of the variable.

The input for CSTEM is contained on several different input decks for each analysis discipline so that separate analysis disciplines may be turned on or off without requiring a large modification of one common input deck. The control of the analysis is achieved mainly from the structural input deck, NT2. This deck contains the geometry information for the model as well as control variables that will activate the various analysis modules of CSTEM.

Free format input is assumed. In most cases, input is read using a general read routine (READZR) which reads each line of input first as an 80 character string. For this reason, **input lines should be no more than 80 characters long**. After reading the input line, the fields contained in the string are then deciphered, with a maximum number of 25 numeric fields and 14 character fields. Field delimiters are blank spaces or commas. Ending an input line with an ampersand (&) will cause the next line of input to be read as a continuation of the line ending with the ampersand. This allows a means for input sequences longer than 80 characters to be input, but the maximum number of fields allowed still applies. The read routine recognizes a dollar sign (\$) as a comment designator so that anything following a dollar sign will be ignored.

The input fields are first assumed to be numbers. The 25 numeric fields are initialized as zero, then filled with the deciphered numbers read. Exponential formats are recognized and the E or D in the exponent is not case sensitive. A blank space following the exponent designator is interpreted as a positive sign for the exponent itself.

If the field is not recognized as a number it will be returned as a character string. In general, there is a maximum string length of 60 characters. The character strings remain in the same case as read so that string matching is case sensitive. Title lines at the beginning of the

individual input files are read and returned as 80 character strings. These title lines should not begin with a number alone unless all input lines contain a separate line number.

The various input files for CSTEM are summarized here. The detailed descriptions and input sheets for these files follow.

<b><u>FILE</u></b>	<b><u>SUMMARY</u></b>
2	ANALYSIS CONTROL, GEOMETRY, STRUCTURAL LOADS AND BOUNDARY CONDITIONS
7	PREVIOUSLY SAVED ELEMENT STIFFNESS
8	PREVIOUSLY SAVED INT.PT. STRESSES ANDD STRAINS
12	STRUCTURAL RESTART FILE
16	DAMAGE RESTART FILE (USER STATE VARIABLES)
22	PREVIOUSLY SAVED INTEGRATION POINT DATA
32	HEAT TRANSFER ANALYSIS CONTROL, LOCATIONS AND LOADS
33	HEAT TRANSFER RESTART FILE
48	DAMAGE MECHANICS STATE RESTART FILE
52	ELECTROMAGNETIC ANALYSIS CONTROL, LOCATIONS AND LOADS
54	ACOUSTICS ANALYSIS CONTROL AND LOADS
75	TAILORING CONTROL
91	(SLOP) NODES TO USE FOR MODESHAPE SLOPE CALCULATION
27	ICAN CONSTITUENT PROPERTIES DATA BANK
50	ABSORPTION MATERIAL DATA BANK

## 4.1. ANALYSIS CONTROL AND STRUCTURAL INPUT

The analysis control and structural input is the primary source of control of a CSTEM analysis. This input file is file code 2, known as the variable NT2 in the CSTEM code. Whether a structural solution is desired or not this file is necessary to define geometry and to initiate the desired analyses. This section contains a general description of the input and how it is used. Section 5.1. contains an explicit summary of these input variables and the order in which they are occur in the input deck. The summaries in Section 5. are referred to as the input sheets and would normally be the pages used by someone actually assembling an input deck.

### 4.1.1. ANALYSIS CONTROL

There are two places in the input deck where information which controls the subsequent reading of the input deck and/or the analyses performed in the run. The first place is at the very beginning of the deck, which is described in Section I. of the input sheets. These variables remain unchanged throughout the entire analysis and contain information about the model size, how the geometry is defined, control of analysis restart, and activation of various analysis modules. The second control information location is at the start of each load case. This information is described in Section XI. of the input sheets. These variables control changes in material properties, load conditions, specific analysis options, and printed output.

#### 4.1.1.1. Initial Control Lines

The initial control lines are the first four lines in the CSTEM input deck. They contain global parameters which do not change throughout the course of the analysis.

##### 4.1.1.1.1. Title Line

The first control information line is for identification of the input deck. An 80 character title will be read and written to the main input echo and output files. This line is examined to determine whether the entire input deck is line numbered or not. An attempt is made to read a single number from the beginning of this first line. If no error occurs it is assumed that the input deck does contain line numbers at the beginning of each line. If an error occurs, as would be the

case if characters are encountered at the beginning of the title line, then it is assumed there are no line numbers throughout the input deck. Therefore, if there are no line numbers the title must begin with at least one non-numeric character in the first word.

#### 4.1.1.1.2. Problem Size Line

The first three variables of the second line are sizing variables. The variable NN indicates the number of nodes, NELTYP is the number of different types of elements, and MMAT is the number of different materials in the model. When generating geometry, if the number of nodes indicated on this line does not agree with the number of nodes after mesh generation the generated geometry will be printed and all further execution is stopped. A negative value of NN signals that geometry will be contained both on the input deck and generated internally. This option allows mesh generation runs only to be performed since NN will not agree with the total number of nodes after mesh generation. Once the mesh generation output has been obtained, the analysis can be performed by changing the input so that the entire geometry is contained on the input deck.

Following these three sizing variables is the number of load cases indicator, NLC. This variable determines how many load cases of the input deck will be read and analyzed. A zero indicates that execution is to stop once the element stiffnesses have been calculated. This calculation can be very time consuming for large layered models. When first running a large layered model it may be desirable to compute the element stiffnesses, save the element stiffness file and integration point file, and stop to check the amount of time required in generating element stiffnesses.

A negative value for NLC indicates that no structural analysis is to be done. This occurs if only a heat transfer or electromagnetic analysis is desired. A micromechanics analysis can be performed following the heat transfer analysis; however, the only loading would be due to the internal thermal mismatches. No structural boundary conditions would come into effect. An additional use for a negative NLC is in the case where a structural analysis has been done previously and the integration point constitutive file, NT8, has been saved. This file contains stress and strain information at each integration point. Additional analysis using the micromechanics



portion of the code, such as cross section interpolation, can be performed without any further structural finite element analysis. The integration point constitutive file must be made local to the job with the file name FILE8 in order for the program to access it.

Following the number of load cases, the geometry definition flag, INGEOM, indicates whether the geometry is contained in the input deck, is to be generated internally, or is contained on separate files. The specific type of the node, element, and layering inputs used depend on the value of this variable. The input sheets divide these input types into different sections with a note at the beginning of each section describing the value of INGEOM that requires that particular input to be used.

The flag indicating whether nodal banding is to be done, NBAND, is next. Unless the model has been banded by some external preprocessor it is a good idea to indicate that banding be done. The results of the banding operation are the first thing on the structural results file, FILE4. The node ordering with the smaller banding criteria will be applied to both the structural and heat transfer portions of the code. The default criteria (used if NBAND is positive) is the profile. By indicating a negative value for NBAND, the criteria can be changed to the absolute value of NBAND, where 1= RMS wavefront, 2=bandwidth, 3=profile, 4=MAX wavefront. Both the original and banded parameters are indicated with an underline below the ordering used.

Following this is a flag, IBLK, which indicates whether to size solution blocks using all of available blank common or only half. This option may be necessary in certain analyses in which two different matrices must be operated on at the same time. Generally, this flag should remain zero unless an execution is stopped indicating that this flag needs to be set to 1. CSTEM has been used mainly for stress recovery only, where all displacements are obtained from some other analysis and input as prescribed displacements. In this case, the solution phase is not necessary and can be avoided by setting IBLK to -1.

The last flag on this line, KADD, indicates whether additional stiffness terms will be input. Care must be taken when using these additional stiffness terms as this feature has not been extensively checked out. Generally, this flag will be zero.

#### 4.1.1.1.3. Restart and Restart File Creation Options

The third line controls creation of a file, FILE10, for later restart of the structural analysis and indicators for activation of an analysis restart. A single load case or all load cases analyzed can be saved on a file for later use in restarting the analysis. The variable NOUT indicates whether this file is to be written or not. The sign of NOUT indicates whether the restart file is to contain all load cases ( $NOUT < 0$ ) or just the latest converged load case ( $NOUT > 0$ ). This variable also controls whether the restart file is an ASCII file ( $|NOUT| < 10$ ) or a binary file ( $|NOUT| = 10$ ). Using NOUT in a restart run, it is possible to read one type of restart file (ASCII or binary) and write another type (binary or ASCII). When performing a restart run (i.e.  $NRESTA \neq 0$ ) setting  $NOUT = 3$  will indicate that an ASCII file is to be read to set up the state from which the analysis is to be restarted, while a binary file for restart will be written as the analysis proceeds. Setting  $NOUT = 30$  does just the opposite (reads binary file, writes ASCII file). This option can be used to generate one restart file type from another type by setting NOUT appropriately with  $NLC = NRESTA$ . The restart file basically contains displacements and, in a nonlinear analysis (large displacement, plasticity, creep, damage), integration point stresses and strains. The geometry, material properties, and loading information for a restart run is obtained from the input deck, but the material state is obtained from the restart file. The results from which the restart proceeds can be written to the printed output file, FILE4, by entering NOUT as 2 or 20 (instead of 1 or 10).

The second variable, NRESTA, indicates whether the analysis is a restart run and from which load case to proceed on the restart file. If a load case number corresponding to the value of NRESTA is not found on the restart file, the analysis will not proceed.

The third variable, INREST, indicates from which load case on the input deck the restart run is to proceed. The input is read in sequential order until the load case corresponding to INREST is found. All geometry, material properties, boundary conditions and revisions of such for load cases previous to load case INREST will be stored as they are encountered so that the input state of the model is the accumulated sum of all the previous load cases on the input deck. The restart analysis begins with load case INREST and proceeds through load case NLC.

For a restart of a damage mechanics analysis, the above variables are input as described. However, in addition to reading the usual restart FILE10 another file containing the damage state must be provided. This file is FILE48 from a previous damage analysis, which contains the damage state for the last converged load case of the previous damage analysis.

Restart of an analysis should not begin with a linearly ramped load case since initial load conditions of zero will be used. Restart of an analysis with linear load ramping may require that the load case read from the restart file be resolved with ramping turned off before continuing with the next ramped load case.

As mentioned previously, the element stiffness and integration point files can be saved and reused when conditions are such that the stiffness does not change. This is a separate activity from the analysis restart. If these files are available the stiffness will not be calculated, if the files are not available the stiffness will be obtained by calculation.

The fourth variable on this line is IPAT. This variable controls the output of PATRAN 2.5 results files. The PATRAN results files will be printed in ASCII form if  $IPAT = 1$  and in binary form if  $IPAT = 2$ . The PATRAN results printed are described in Section 6.19.

The fifth variable is ICHK. This variable is used to perform a check run of the first load case on the input deck, including printing of the layer visualization file described in Section 4.1.4.5. If  $ICHK = 1$  is entered, the first load case is read and written to the echo file, and the layer orientations are calculated and will be written out if requested by the variable LYPRNT.

#### 4.1.1.1.4. Analysis Options Line

The fourth line contains analysis control variables. The first variable, IPLO, is the global plasticity analysis flag. This flag is used to indicate that plasticity analysis will be performed and appropriately sizes material file record sizes. Plasticity analysis is turned off for  $IPLO = 0$ . The sign of IPLO indicates the nonlinear solution method to use, initial stiffness ( $IPLO = 1$ ) or tangent stiffness ( $IPLO = -1$ ).

The second variable, ICRO, is the global creep flag. A value of ICRO = 1 indicates that a creep analysis is to be performed and activates appropriate sizing of material file record sizes. Creep analysis is turned off for ICRO = 0.

The third variable, IDIS, is the geometric nonlinear analysis flag. A small displacement analysis (IDIS = 0) or an updated Lagrange large displacement analysis (IDIS = 1) are the only options available. A total Lagrange large displacement analysis is currently not implemented. Normally, all stress and strain output is referenced to the undeformed configuration. By indicating a large displacement analysis as a negative, the stress and strain output will be referred to the deformed configuration.

The next two variables apply to large displacement analyses only. If a small displacement analysis is indicated they will be ignored. These variables are ITCVG, which indicates the maximum number of equilibrium iterations allowed per load step, and CVGTOL, the tolerance used in determining convergence for equilibrium iterations. This tolerance is checked against the maximum displacement increment from one iteration to the next. The sign of ITCVG determines the action taken after the maximum number of equilibrium iterations has been reached, whether to print the current results and stop or continue execution.

The sixth variable on this line, IHTR, activates the heat transfer module. This indicates that there will be a heat transfer analysis to be done at some point in the current run. An actual heat transfer analysis need not be done in each load case, as indicated on the load case control line. The sign of IHTR determines the type of units used in the heat transfer inputs.

The seventh variable, IDMO, is the global damage mechanics analysis flag. Damage analyses are turned off for IDMO = 0. An initial stiffness iteration method is used for damage analysis for IDMO = 1 and a tangent stiffness method is used for IDMO = -1.

The eighth variable, IEMAG, activates an electromagnetic analysis and indicates the type of method to be used: WAVES (IEMAG = 1), single reflection light ray (IEMAG = 2), or experimentally generated absorption tables (IEMAG = 3). As with the heat transfer analysis, a variable on the load case control line can turn the analysis on and off for specific load cases.

The ninth variable, LDORDER, pertains to the large displacement option. It indicates whether or not to include the higher order terms in the calculation of strain and its transformation from one reference configuration to another. If set to zero or not specified, the default is to not include the higher order terms, LDORDER = 1. They are included if LDORDER = 2.

The tenth variable on this line, NOISE, controls the acoustic analysis module. An acoustic analysis must be done in conjunction with an eigenanalysis, but an eigenanalysis can be performed without doing an acoustic analysis. Setting this flag to one indicates that an acoustic analysis is desired for every load case in which an eigenanalysis is done.

#### 4.1.1.2. Load Case Control Lines

There are two load case control lines which control loading types to be read, material properties to be read, specific analyses to be performed in the load case, and the results to be printed for the load case. These lines are described in Section XI.1. of the input sheets.

##### 4.1.1.2.1. First Load Case Control Line

The first variable indicates the rotational speed of the model in RPM. The second variable, IAXS, indicates the global axis about which the rotation is occurring (1=X, 2=Y, 3=Z). The third variable, IACC, indicates whether acceleration loads are to be read in the current load case as described in Section XIII. of the input sheets. These load types can conceivably change from non-zero to zero values, so that a check for a zero value cannot be used to bypass calculation of these loads. For analyses in which these loads are never applied, the rotation axis and acceleration indicator can be set to -1 so that the calculations involving these load types are bypassed, thereby saving time.

The fourth variable, IELA, indicates whether material properties are to be changed in the current load case. Common elastic material properties are read if IELA =  $\pm 1$ , as described in Section XIV. of the input sheets. A value of IELA=0 would indicate that no material properties are to be read. However, for the very first load case of an analysis (and only the very first load case), a value of IELA=0 will be interpreted in the same way as IELA=1, since material properties are required to perform an analysis. A positive value (IELA = +1) indicates that a table of material

properties will be read from the input deck. A negative value ( $IELA = -1$ ) indicates material properties will be calculated by the micromechanics module and only the material specification codes and parameters will be read from the input deck.

User specification of any type of material properties, including those which are entirely unknown to CSTEM, can be input using the UMATIO routines. The UMATIO routines can be accessed by indicating  $IELA=2$ . The UMATIO routines are used with user plasticity, creep, and damage models as explained in Section 3.1. 11. 2.4. The IELA variable can be packed to indicate that both the common method of material input ( $IELA=\pm 1$ ) and the user method ( $IELA=2$ ) are to be used by entering  $IELA=12$  or  $IELA=-12$ .

The fifth variable, IPC, indicates changes in the inelastic analysis material properties are to be read from the input deck. This is used with the plasticity, creep, and/or damage options implemented in CSTEM. IPC is a packed flag, with up to three integers forming the single IPC value. A 1 indicates plasticity properties are to be read, a 2 indicates creep properties are to be read, and a 3 indicates damage properties are to be read. Packed combinations (12, 123, 23, etc.) are used to indicate that more than one type of inelastic property is to be read.

The sixth variable, IEIG, is used to indicate the number of eigenvalues and vectors to calculate for a free vibration analysis. The maximum number of eigenvalues and vectors that can be solved for is indicated by the parameter MXROOT. If an acoustics analysis is requested in the initial control lines, it will be performed whenever an eigenanalysis is done. The sign of this variable indicates whether to use the subspace iteration technique ( $IEIG > 0$ ) or the determinant search technique ( $IEIG < 0$ ) to calculate the eigenvalues and vectors.

The seventh variable, IHT, is the heat transfer indicator. This variable determines the type of heat transfer analysis that will be performed (1 = linear steady state, 2 = nonlinear steady state, 3 = linear transient, 4 = nonlinear transient). The input for the heat transfer analysis is contained on a separate file, NT32. In this way the heat transfer analysis can be turned on (or off for  $IHT = 0$ ) by simply changing the value of this variable. For cases in which a transient heat transfer load case spans multiple structural load cases, a negative value is used to indicate a continuing heat transfer load case so that no additional input will be read from the heat transfer deck.

The eighth variable, IBUCK, indicates whether a buckling analysis is to be performed, and how for how many modes. The limit on the number of modes is the same as for the free vibration frequency eigenanalysis. The sign of this variable determines whether the original configuration stiffness ( $IBUCK > 0$ ) or deformed configuration stiffness ( $IBUCK < 0$ ) will be used.

The ninth variable, LABSO, is the secondary electromagnetic analysis indicator. This variable turns the electromagnetic analysis on or off for the current load case. The method used is indicated at the beginning of the input deck on the initial control lines by variable IEMAG. The input to the electromagnetic analysis module is contained on a separate input file, NT52.

The tenth variable on this line, ICAN, controls the micromechanics analysis module. A full micromechanics analysis can only be done if the material properties have been calculated using the micromechanics module. Once a structural finite element solution has been calculated, the results are interpolated to individual cross sections specified in Section XXV. of the input sheets and processed to form the necessary loading inputs for the ICAN micromechanics program. These loading inputs are either displacements ( $ICAN = 1$ ) in the form of reference plane strain and curvatures or loads ( $ICAN = 2$ ) in the form of stress and moment resultants. Individual ply stresses and strains are then calculated from these inputs as detailed in the ICAN manual. It is possible to use the finite element interpolated ply strains ( $ICAN = -1$ ) or both interpolated ply strains and stresses ( $ICAN = -10$ ) rather than those calculated by ICAN from the input loadings. The micromechanics analysis results are printed to FILE61.

The micromechanics module can also be used to simply interpolate the finite element ply strains or stresses to a specified cross section. In this case the material properties may be either calculated using micromechanics or input manually. This option can be useful for obtaining nodal quantities to use for contour plots. The value of the ICAN variable determines whether the results of this interpolation are printed with respect to the ply material coordinate system with 11 in the fiber direction ( $ICAN = 3$ ), the global coordinate system ( $ICAN = 4$ ), or in an unrotated material coordinate system, so that 11 lies along zero degrees regardless of ply orientation ( $ICAN = 5$ ). The interpolated stress results can be printed to FILE61 in a general printed output form ( $ICAN > 0$ ) or

in a SIESTA layered UOF ( $ICAN < 0$ ). Note that these print options apply to cross section interpolation only,  $|ICAN| = 3, 4, \text{ or } 5$ .

The eleventh variable on this line, IFC, controls whether a macro level integration point failure criteria will be calculated. Turning on this indicator will result in the reading of failure strengths and parameters as described in Section XIV.4. of the input sheets, with the appropriate calculations performed during stress recovery at each integration point. Note that the failure criteria is applied using stress (or strain) values oriented as requested for printout by the variable IPOUT.

#### 4.1.1.2.2. Second Load Case Control Line

The first variable on the second load case control line, NDIS, indicates how many nodal prescribed displacement lines described in Section XIX. of the input sheets are to be read for the current load case. These nodal displacements are generally non-zero values, or will be non-zero at some point in the analysis. Degrees of freedom which remain zero throughout an analysis are more efficiently handled as nodal fixities. Once a nodal displacement is applied, it retains its value until explicitly changed in a subsequent load case.

The second variable, NFOR, tells how many nodal force inputs are to be read in the current load case. These inputs are described in Section XX. of the input sheets. Once a nodal force is applied, it retains its value until explicitly changed in a subsequent load case.

The third variable, NTEM, indicates how many nodal temperature inputs are to be read for the current load case. These nodal temperatures are also used as initial nodal temperature values for a heat transfer analysis. The nodal temperature inputs are described in Section XXI. of the input sheets. All nodal temperatures are initially set to zero degrees if not otherwise specified. As a timesaving feature, NTEM can be set to -1 to bypass thermal load calculations in those cases where there is no thermal loading.

The fourth variable, NPRE, indicates how many elemental pressure inputs are to be read for the current load case. These inputs are described in Section XXII. of the input sheets. Once pressures are applied, they retain their value until explicitly changed in a subsequent load case.



The fifth variable, NADD, indicates the number of element addition or removal input lines to read in the current load case. Besides adding or removing an element, this feature can be used to change the material type of an element. These inputs are described in Section XXIII. of the input sheets.

The sixth variable, IPOUT, pertains to the type of printed output desired for the current load case. The types of output are nodal displacements, nodal reaction forces, and stress/strain output. The default for this variable (IPOUT = 0) will indicate that all nodal displacements and reactions are to be printed, and the stress/strain results are to be printed in the global coordinate system at every integration point of each element using KSI units for stress. A summary of the max and min global stress and strain components are printed as well. All output for the load case can be suppressed (IPOUT = -1). Any combination of the default output can be activated or suppressed by the use of a packed code, where the different places (ones, tens, hundreds, thousands) of IPOUT control different output results.

The ones place of IPOUT controls print of displacements, which can be turned off (=0) or on (=1). For very large or small displacement values, an E format can be requested (=2). A summary of displacement max and mins only can be requested (>2).

The tens place of IPOUT controls print of reaction forces, which can be turned off (=0) or on (=1). A summary of reaction force max and mins only can be requested (>1).

The hundreds place of IPOUT controls the stress/strain printout. Stress/strain printout can be turned off (=0), printed in ksi units with respect to global (=1), material (=2), or principle (=3) orientations, or printed in psi with respect to global (=4), material (=5), or principle (=6) orientations. A couple of special forms of stress/strain printout are also available. A mission form (=8), developed for use with MRLIFE residual strength calculations, has a limited number of column labeling banners and outputs integration point stresses with temperatures on the same line. An in-plane principle stress/strain option is also available which finds the principle stresses in the material 1-2 plane and the associated angle about the material 3 axis. The two interlaminar shear stresses (23 and 13) are rotated so that only the 13 stress is non-zero, thus providing a maximum interlaminar stress value along with the associated angle.

Finally, the thousands location of IPOUT controls the stress/strain summary printout. If detailed stress/strain print is requested (hundreds place  $\neq 0$ ), the stress/strain summary is automatically performed in the same orientation. It is only if a stress/strain summary with no detailed stress/strain output is desired that this place is of use. In such as case, the option is to print the summary in global (=1) or material (=2) orientations. For very small effective stress values, the max effective stress will automatically be written in psi and noted as such on the output.

The seventh variable, TIME, is used in creep analyses. TIME is the time increment in hours covered by the load case. It is not the ending time of the load case. The load case ending time would be the sum of all TIME values entered in the current and all previous load cases. For load cases without creep it can be used as a convenient way of labeling load cases. This variable has no relation to the time variable used in a transient heat transfer load case.

The eighth variable, NTIMND, is used to control when a structural analysis is performed during a transient heat transfer analysis. NTIMND refers to one of the NTIMSP time step points, TIMSP, input to a transient heat transfer analysis as described in Section XXVII.3. of the input sheets. At the time TIMSP(NTIMND) in the transient heat transfer analysis, the nodal temperature results are transferred to the structural module and a solution of the structural load case is calculated before continuing with the heat transfer solution.

The ninth variable on this line, NFAIL, indicates how many failed stiffness lines as described in Section XXIV. of the input sheets are to be read for the current load case. These failed stiffnesses are integration points which are not to be included in the element stiffness calculations, thus reducing the stiffness of the element.

The tenth variable, IEIGST, controls whether or not modeshape stresses are calculated. If a value of 1 is indicated, stress and strain values for each requested vibration mode are calculated and printed for only the top and bottom ply of each element. A value of 10 for IEIGST means that an entire stress/strain printout is generated for all layers of an element for each modeshape. For unlayered elements, a nonzero value for IEIGST results in a complete stress/strain printout. For values of IEIGST > 10, an entire stress/strain printout is generated for all layers of an element for the first (IEIGST - 10) modeshapes. The form of the output is similar to the sstatic stress/strain

results printout. If IEIGST is entered as a negative number, the form of the output is a SIESTA layered UOF.

The eleventh variable, NEOUT, is a flag used to indicate selective output of stress/strain results. If NEOUT is zero (or omitted) stress and strain results are printed for all elements. If NEOUT is not zero, stress and strain results will be printed for only the first NEOUT elements as determined from the input order of elements on the input deck. Remember that the element ordering on the input deck does not have to be sequential. For example, if the stress/strain results for elements 5 and 10 are the only ones desired, set NEOUT=2 and put the definitions for elements 5 and 10 as the first two elements in the element definitions (Section V.) on the input deck.

#### 4.1.1.3. Keyword Input

Keyword input is supplemental input which may or may not exist in any particular input deck. Keyword input must be entered immediately following the load case control lines, but may be entered in any order. This type of input contains a 4 letter keyword as the first item on the input line. The keyword indicates the type of input contained on the line. Keyword input is read until no more keyword lines are found, at which time the structured input continues to be read as usual. The currently recognized keywords are CDBG, DACC, DDBG, EIGS, LDBK, MATX, METH, MICR, NRIT, PDBG, RDKS, and VIBC. Most of these keyword inputs turn off or revert to default values for any load case they are not explicitly entered. The exceptions are DACC, LDBK, MATX, METH, and NRIT, which retain assigned values until explicitly changed in a subsequent load case.

##### 4.1.1.3.1. Debug Printout Controls, PDBG, DDBG, and CDBG

Three of the keyword inputs control debug printout in their respective analysis modules. These are the creep debug keyword, CDBG, the damage debug keyword, DDBG, and the plasticity debug keyword, PDBG. Printout is generally written to the file NTDBG. A single integer value following the keyword indicates the internal element number for which debug output will be printed. Often, only debug output for the first integration point in the element will be printed.

#### 4.1.1.3.2. Eigenanalysis Solution Parameters, EIGS

In general, the default parameter values for the subspace iteration method of computing eigenvalues are sufficient to arrive at a converged solution. However, there may be instances when it might be desirable to change the values of these parameters. This line can be used to make these changes. The input line will not be interpreted as such unless the upper case character string EIGS is the first entity contained on the line. Section XII.1. of the input sheets describes the parameters that can be changed. Any parameter interpreted as 0 will be assigned the default value.

#### 4.1.1.3.3. Large Displacement Buckling, LDBK

The large displacement buckling keyword, LDBK, can be used to eliminate the stress stiffness matrix in a large displacement analysis and only the change in geometry will contribute to any change in stiffness. Buckling can be predicted by examining subsequent load case displacements rather than with linear eigenvalue buckling. A single integer value following the keyword is read, with large displacement buckling activated if this integer is non-zero. Once turned on, this option remains in effect until explicitly deactivated by entering LDBK 0.

#### 4.1.1.3.4. Nonlinear Force Calculation Method, METH

The nonlinear force calculation method keyword, METH, is used to indicate how the pseudo force is to be calculated for an initial stiffness (right hand side) nonlinear solution method. This applies to plasticity, damage, and creep analyses. A single integer value following the keyword is read. For a value of 1, the pseudo force is calculated as the residual between applied and internal forces. For a value of 2, the pseudo force is calculated directly from the inelastic strains. This is the default method used if the METH keyword is not entered. Once specified, the value assigned remains in effect until explicitly changed in a subsequent load case.

#### 4.1.1.3.5. Newton–Raphson Iteration Method, NRIT

The Newton–Raphson iteration method keyword, NRIT, indicates whether modified or full Newton–Raphson is to be used in a tangent stiffness nonlinear solution method. This applies to

plasticity and damage analyses. A single integer value following the keyword is read. For a value of 0, the default modified Newton–Raphson is used, with the stiffness matrices updated only at the beginning of a load case. A non-zero value indicates the number of iterations before the stiffness should be updated so that a value of 1 will indicate full Newton–Raphson iteration, with the stiffnesses updated after every iteration. Once assigned, this option will retain its value until explicitly changed in a subsequent load case. It is recommended that this option not be used and the default modified Newton–Raphson be accepted.

#### 4.1.1.3.6. Damage Acceleration, DACC

This option is only applicable to damage mechanics analyses using the initial stiffness method ( $IDMO > 0$ ) in conjunction with the  $METH=2$  keyword option. The  $METH=2$  option is the default for initial stiffness damage mechanics. A single positive integer value following the DACC keyword is read. This value indicates the number of iterations between acceleration calculations. The Aitken  $\Delta^2$  method is used to accelerate the effective damage pseudo stress to the converged value. If a zero is entered, acceleration is turned off. Any positive value other than zero activates the acceleration option and sets the iteration interval for acceleration to that value. The default value for this option is 3 so that acceleration of the solution will occur every 3rd iteration. This keyword option is retained from load case to load case until explicitly changed.

#### 4.1.1.3.7. Damage Iteration Averaging, DAVG

This option applies to damage mechanics analyses only. A nonlinear iterative analysis such as damage mechanics may sometimes exhibit oscillatory or possibly divergent behavior from iteration to iteration. This behavior may be determined by examining the maximum convergence criteria values for each iteration, printed to the timing summary. One method for damping out such oscillations is to average results between iterations. The DAVG keyword allows the user to control averaging of the elastic matrix,  $[D]$ , and the damage strain,  $\{\epsilon\}_d$ , over the current and previous iteration. A single positive integer value following the DAVG keyword is read. A value of 0 indicates no averaging (the default). A value of 1 indicates that the elastic matrix,  $[D]$ , is to be averaged, but the damage strain is not averaged. A value of 2 indicates the reverse; the damage

strain,  $\{\epsilon\}_d$ , is averaged, but not the elastic matrix. A value of 3 indicates that both the elastic matrix and damage strain are to be averaged. The value associated with this keyword option is retained from load case to load case until explicitly changed.

#### 4.1.1.3.8. Read Alternate Stress Stiffness, RDKS

The RDKS option allows the use of a different stress stiffness matrix for linear eigenvalue buckling analyses than that generated for the current set of boundary conditions. This may be desirable for certain panel buckling problems, in which pinned edge supports are applied together with in plane edge loads. A single integer value following the keyword is read. A normal buckling analysis is indicated by a zero value, in which the geometric and stress stiffnesses calculated for the given boundary conditions are used in the buckling analysis. For a value of 1, the stress stiffness is generated for the given boundary conditions, written to FILE7, and execution is halted. This generates the alternate stress stiffness matrix. For a value of 2, the stress stiffness is read from a saved FILE7 and used with the calculated geometric stiffness in the buckling analysis. The user will be prompted to enter the FILE7 filename.

#### 4.1.1.3.9. Change Vibration Boundary Conditions, VIBC

The VIBC option activates a read of additional prescribed displacement input lines just prior to eigenvalue frequency calculations. This allows static solutions to be performed with one set of prescribed displacements and frequencies calculated for a modified set of prescribed displacements. This can be useful to avoid rigid body static solutions when using additional stiffnesses as system stiffnesses to simulate system frequency solutions. A single integer value following the keyword is read, which indicates the number of additional prescribed displacement lines to be read.

#### 4.1.1.3.10. Microstress Plot Output, MICR

This option creates output which can be processed to generate fiber and matrix stress contour plots. This option is only available in conjunction with a full ICAN analysis (variable ICAN = 1, 2, -1, or -10, see Section 4.1.1.2.1.). Version numbered files PATMICROF,

PATMICROA, and PATMICROB are created which contain microstress results for each ply at requested ICAN analysis locations. ICAN analysis at all layered nodes (NCS= -2) should be requested to enable contour plotting. An external program, layavg.x, is available at GEAE to complete the processing of these microstress files.

#### 4.1.1.3.11. Material Selected Stress Summary, MATX

The stress/strain summary printed to the output file is usually the max and min component values found in the model without regard to material type. This option allows up to 3 different materials to be selected for a separate summary of max and min stress values. (The limit of 3 is set by a PARAMETER, JMAT). If less than 3 materials are selected, a summary of max and min stress over the entire model is also printed. If 3 materials are selected, the overall summary will be for the combination of just those three materials. This option will remain in effect from load case to load case until explicitly changed. Entering the keyword MATX with no material numbers deactivates the option.

#### 4.1.1.3.12. Linear Load Ramping, RAMP

The capability exists to ramp loads from one load level to another in any number of equal load steps. Linear load ramping is activated using the keyword RAMP, with an integer number of equal load steps indicated. Nodal forces and displacements, pressures, RPM, accel and thermal loads are ramped. Stiffnesses are not ramped. Stiffnesses at the final temperature value are used throughout the load case. Results will be output at the final load step only, not at intermediate load steps. The linear ramping option must be reactivated for each load case for which ramping is desired. A value of 0 or 1 indicates no ramping.

Restart of an analysis with ramping must be initiated with an unramped load case so that the initial load conditions for ramping can be established. This may require resolving the load case read from the restart file before continuing with the next ramped load case.

#### 4.1.1.4. File SLOP Mode Shape Slope Input

Mode shape slope calculations are performed for each vibration frequency requested if a file with the name SLOP (all uppercase) is found local to the CSTEM run. The presence of SLOP is the only necessary item to initiate this calculation. Each line of this file corresponds to a particular span location for which the modeshape slope is to be calculated. The five parameters on each line are NLE, NMID, NTE, IAXI, and ITAN. These parameters are the number of the node whose deflections are to be used for the leading edge deflections (NLE), the node whose deflections are to be used for the midchord deflections (NMID), the node whose deflections are to be used for the trailing edge deflections (NTE), the degree of freedom to be used for the axial direction (IAXI), and the degree of freedom to be used for the tangential direction (ITAN).

#### 4.1.2. GEOMETRY

The CSTEM code uses isoparametric bricks containing 8, 16, or 20 nodes. The 8 node bricks use linear shape functions, the 20 node bricks use quadratic shape functions, while the 16 node bricks use quadratic shape functions in 2 directions and linear in the third. The 20 node bricks using reduced (2nd order) integration are the recommended element for use in a general problem due to its greater flexibility and higher order displacement capability. Next would be the 8 node brick, with incompatible modes activated to allow more accurate modeling of bending.

##### 4.1.2.1. Nodes

Nodes are points in space which are specified by a unique node number and the three coordinates of the node in the global coordinate system (X,Y,Z). Both structural nodes and reference nodes are used in CSTEM.

Structural nodes are connected into elements by the assumed displacement equations of the particular element type. Section II. of the input sheets pertains to structural node input. The system of equations relating the global degrees of freedom of the entire structure are set up based on the order of input of the structural nodes, which means that the nodes do not have to be numbered in consecutive increasing order. An option to reorder the set of equations to give the smallest nodal bandwidth (and thus a quicker solution) is available. There is a limit on the number of structural



nodes allowed (NNMAX) as well as on the largest structural node number allowed (NO). These limits are often, but not necessarily, the same and are checked on input.

Reference nodes are points in space which are used to set up transformations as described in Section VIII.1. of the input sheets. Nodes which are reference nodes only are temporary and are input as described in Section VIII.1. of the input sheets. These nodes are not retained once the transformation matrix has been calculated; therefore, reference nodes are not subject to any nodal limits as are structural nodes. Structural nodes can be used as reference nodes in defining the transformations of Section VIII.1. In fact, structural nodes are examined first when forming a transformation with this method. If a structural node with the specified reference number is not found, the reference node input is examined to find the specified reference number. For this reason reference nodes must have node numbers which are different from structural nodes.

#### 4.1.2.2. Elements

An element is the 3D volume in which a certain displacement pattern which relates the node points of the element to all other points in that volume is assumed. Section V. of the input sheets pertains to element input. The elements are specified on input by a unique element number, the node numbers of the nodes associated with the element, and the material type in the case of homogeneous material models. The nodes connected to the element must be input in a particular order, which defines the structural coordinate system for that element. This node input order (called connectivity) also defines the numbering of the 6 element faces, which provides a convenient way to visualize the orientation of the structural coordinate system for an element. Figure 3. shows the relationship between the element connectivity, the structural coordinate system, and the element faces. The element numbers do not need to be input in consecutive increasing order, but there is a limit on the number of elements (NEMAX). There is no specific limit on the largest element number.

Elements may be made up of a single material type and orientation or layered with multiple material types and orientations. When the element is composed of material layers some extra consideration must be given to the shape and the nodal connectivity of the element since the layers are often referenced to the structural coordinate system of the element. The element shapes should

be such that, within each element, at least one axis of this coordinate system is consistently parallel or perpendicular to the actual physical layers. Keep in mind that it is much easier to specify the element layering when the structural coordinate systems of the elements are all lined up in the same direction. Also to ease this specification, the elements should be numbered such that they progress through the structure in a consistent pattern.

#### 4.1.2.3. Mesh Generation

Mesh generation of basic doubly curved shapes, including cylinders, cones, and planes, can be done automatically in CSTEM. Mesh generation is activated by setting the control variable INGEOM to a negative value. Section V.6. of the input sheets describes the input variables for mesh generation. Multiple shapes can be generated and coincident nodes are checked for and replaced by a single node. The magnitude of INGEOM indicates the number of separate surfaces that will be generated. The nodes and elements are numbered in the order that they are generated. The basic input required for generating a meshed surface is to locate the shape in the global coordinate system, define the thickness and the number of elements through the thickness, and specify the radius, curvature and number of element divisions about two orthogonal rotation axes.

To use the mesh generation capability in CSTEM, it is best to have an understanding of the steps followed to generate a surface. The shape is located and the thickness is specified by defining the global location of two points on the structure. One point lies at the center of the inner surface of the generated shape and the other lies at the center of the outer surface of the generated shape. The thickness axis of the structure is set up such that it passes through the global coordinate system origin, the inner surface point, and the outer surface point. The vector distance between the inner and outer surface points is the thickness. Two rotations are specified to generate the shape, which is generated by incrementally sweeping through the rotation 2 angle for each rotation 1 increment and generating nodes through the thickness at each increment location.

A mesh generation coordinate system is set up by rotating the global coordinate system through two Euler rotations as is necessary so that the  $x''$  axis passes through the inner and outer surface points. The  $x''$  axis will thus be the through thickness axis for mesh generation. To set up this  $x''$  axis the global system is first rotated about the global Z axis and then a second rotation is

done about the resulting  $y'$  axis. The resulting coordinate system is referred to as the mesh generation or double prime ( $''$ ) coordinate system.

A rotation 1 and a rotation 2 are specified in the input to generate the shape. These are not to be confused with the two Euler rotations done automatically as described above. Each rotation has an angle, a radius of curvature, an axis of rotation, and a number of elements associated with it. The axes of rotation are the  $y''$  and  $z''$  axes. These two axes and the  $x''$  thickness axis make up the mesh generation coordinate system. The information for either rotation axis can be specified as rotation 1 and the other as rotation 2.

The starting point for the mesh is found by rotating the  $x''$  axis by  $-1/2$  the rotation 1 angle, then  $-1/2$  the rotation 2 angle about the respective rotation axes. Nodes are generated through the starting point thickness according to the number of elements in the thickness direction. The  $x''$  axis is then rotated a positive rotation 2 increment and the nodes through the thickness at this location are generated. The rotation increments are determined by the number of elements specified in the rotation directions. This process continues until the  $x''$  axis has been swept through the entire rotation 2 angle and nodes have been generated through the thickness at each increment. The  $x''$  axis is then rotated a positive rotation 1 increment and back to the  $-1/2$  rotation 2 position and the process is started again. This continues until the  $x''$  axis has been swept through an entire rotation 1 angle.

#### 4.1.2.4. Node / Element Examples

##### 4.1.2.4.1. Basic Input Example

This example is for a 20 node brick made up of a homogeneous isotropic material. Examples of 8 and 16 node elements are subsets of the 20 node element since the same first 8 or 16 nodes would be used. This example is for a single element section of a cylinder. The single element is shown in Figure 21. Figure 22. is a split image in which the upper part contains the actual input to generate the element, and the lower part contains the sections of the input sheets which describes this input. This simple example illustrates the basic input for nodes and elements.

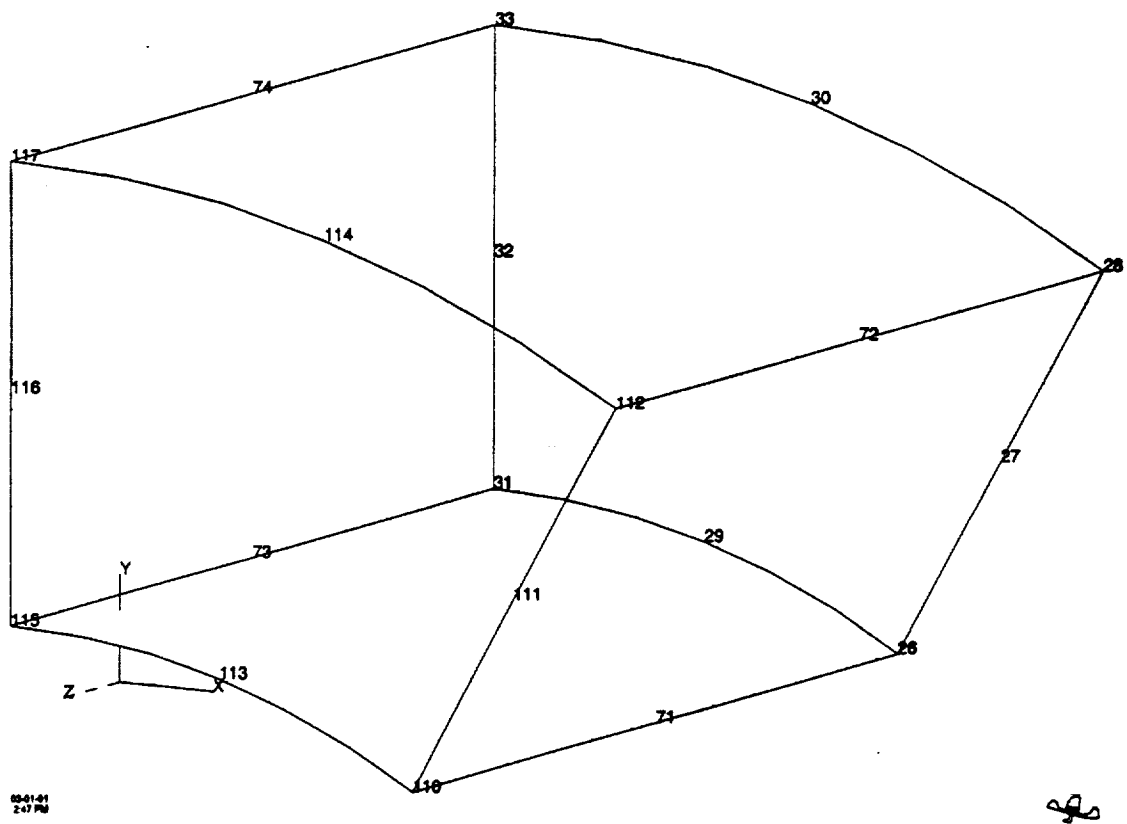


FIGURE 21. SINGLE ELEMENT CYLINDER SECTION

## II. NODAL COORDINATES

N X Y Z

## III. NODAL SKEW COORDINATE SYSTEMS

### III.1. Input Skews

NSKEW

*Enter NSKEW of the following lines.*

N NTRN NEND NINC

## IV. CONSTRAINT EQUATIONS

NCST

## V. ELEMENT DEFINITION

### V.1. Header Line For Each Element Type

NTYPEL NELEMS INCOMP INTORD ISTRP KSAVE LYPRNT

### V.4. 20 Node Solid (NTYPEL = 20)

NEL N1 N2 N3 N4 N5 N6 N7 N8

N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 IMAT

\*\*\*\*\*

```
26 .50000E+00 .86603E+00 -.50000E+00 $ NODAL INPUT FOR 20 NODES
27 .62500E+00 .10825E+01 -.50000E+00
28 .75000E+00 .12990E+01 -.50000E+00
29 .25882E+00 .96593E+00 -.50000E+00
30 .38823E+00 .14489E+01 -.50000E+00
31 .00000E+00 .10000E+01 -.50000E+00
32 .00000E+00 .12500E+01 -.50000E+00
33 .00000E+00 .15000E+01 -.50000E+00
71 .50000E+00 .86603E+00 .00000E+00
72 .75000E+00 .12990E+01 .00000E+00
73 .00000E+00 .10000E+01 .00000E+00
74 .00000E+00 .15000E+01 .00000E+00
110 .50000E+00 .86603E+00 .50000E+00
111 .62500E+00 .10825E+01 .50000E+00
112 .75000E+00 .12990E+01 .50000E+00
113 .25882E+00 .96593E+00 .50000E+00
114 .38823E+00 .14489E+01 .50000E+00
115 .00000E+00 .10000E+01 .50000E+00
116 .00000E+00 .12500E+01 .50000E+00
117 .00000E+00 .15000E+01 .50000E+00
0 $ NODAL SKEWS (For rotated nodal boundary conditions)
0 $ CONSTRAINT EQUATIONS
1 20 0 3 -1 0 0 $ ELEMENT HEADER LINE
6 28 33 117 112 26 31 115 110 $ SINGLE ELEMENT NUMBER 6
30 74 114 72 29 73 113 71 27 32 116 111 1
```

FIGURE 22. BASIC NODE AND ELEMENT INPUT

#### 4.1.2.4.2. Mesh Generation Input Example

This example generates the 50 element double curved surface shown in Figure 24. The input is shown in Figure 23. The elements are 20 node bricks and are all made of the same material, the properties of which are those defined for material #1 in the material property input.

##### I.1. Pregenerated Skews

NSKEWP

##### I.2. Mesh Generation Definitions

###### I.2.a. Thickness Specification

XI YI ZI XO YO ZO NELMT

###### I.2.b. Rotation #1 Specification

ANG1 RAD1 IAX1 NELM1

###### I.2.c. Rotation #2 Specification

ANG2 RAD2 IAX2 NELM2

###### I.2.d. Layer Orientation Definition

NGEN

###### I.2.e. Layer Material Definition

NSET NLAY

MAT ANG THICK

IGEN(I), I=1,N

\*\*\*\*\*

0	\$ pregenerated skews
20 0 0 2 0 0 0	\$ element header line for generated elems
1. 0. 0. 1.75 0. 0. 2	\$ thickness specification
45. 3.0 2 5	\$ rotation 1
90. 0.5 3 5	\$ rotation 2
1	\$ # layer generation sets
1 1	\$ # layers in set
1 0. 0.75	\$ layer material properties
1	\$ generation order

FIGURE 23. MESH GENERATION EXAMPLE

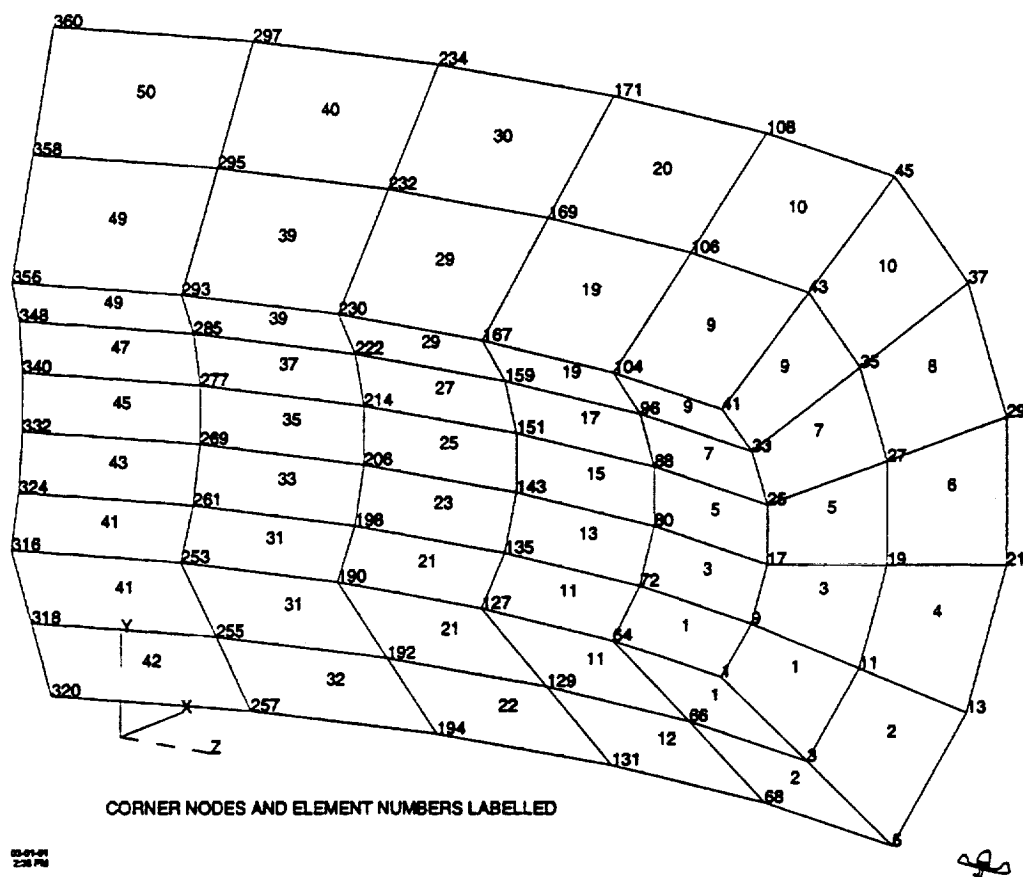


FIGURE 24. GENERATED DOUBLE CURVED SURFACE

#### 4.1.3. ELASTIC MATERIAL PROPERTIES

Elastic material properties are required for any structural analysis performed in CSTEM. Materials used in CSTEM can be either isotropic or orthotropic. Section XIV. of the input sheets pertain to material input. The material properties can be modified from load case to load case. Material properties can be either input in table form or calculated from constituent properties for fiber/matrix composites, but not both. There is also a capability for the user to supply material matrices directly rather than material properties.

For the table input method, each material is specified on input by a unique material number, a stress free reference temperature, the density, and the elastic properties of the material as a function of temperature. The properties must be input in either increasing or decreasing temperature order. There is no limit to the number of materials that can be input to a given problem using the table input method. There is a limit (currently, NTEMP=10) on the number of temperatures at which the elastic properties can be defined.

For the micromechanics calculation option, the constituents, the ratio of these constituents in the specific material system, the cure temperature and the percentage of moisture are specified. The unique material numbers are assigned consecutively in the order of input, with a limit (currently, MXIMAT=10). The constituent properties are obtained from a data bank and the material properties are then calculated for the temperature at the point of interest. The ICAN User Manual, included as an appendix to this manual, contains additional information pertaining to the calculation of the material properties and format of the constituent material property data bank.

Some special features of the material inputs may be useful. For the table input method, the material density is used to calculate rotational loads, weight of the structure, and structural mass used in free vibration analysis. If no rotational loads exist and no vibration analysis is being done, a zero density results in zero weight for that material with no further detrimental effects. If the material properties are specified at only one temperature, those properties are used at all temperatures. Also, a 'zero material' option is available in which a material with  $EX = 0$  will not be included in the structural stiffness. It will be included in the structural mass if the density is not zero.



As a note of caution, when using table input of orthotropic material properties care must be taken to ensure that the Poisson ratios are correctly defined. The CSTEM code uses a row normalized definition of Poisson ratio:

$$\nu_{ij} = \frac{\epsilon_j}{\epsilon_i} = \nu_{ji} \frac{E_i}{E_j}$$

This is not a problem with isotropic materials since the Poisson ratios are the same.

The orientation of orthotropic materials can be specified in several ways. For a homogeneous (unlayered) element, a skewed material coordinate system which transforms the orthotropic material coordinate system to global can be assigned to the element. This is done on input immediately following the element definition by assigning a transformation number to the element. The particular transformation matrix associated with this transformation number is defined later in the input deck using reference nodes, direction cosines, or Euler angles. For layered elements, an additional method for specifying this orientation is available in which the material coordinate system is oriented with respect to the structural coordinate system of the element. This method requires specification of the element layering. This method can also be used for homogeneous elements by specifying the element be layered with a single layer. It should be noted that if a skewed material coordinate system is assigned to a layered element, the orientation specified in the element layering will be used.

#### 4.1.4. LAYERING

Layering is done on an element level and allows a single element to contain several different materials and/or material orientations. There is a limit on the number of layers in an element, set by the parameter LAYMX. Layering may be used directly with layered elements or the layering information may be used to generate equivalent homogeneous orthotropic material properties for use with homogeneous 3D elements. Section VII. of the input sheets pertains to element layering. It is also recommended that Section 2.4.3. of the CSTEM User Manual be read to gain a better understanding of what occurs when layering elements.

There are basically three different ways that the layering of elements can be specified. The first method is a manual way to directly assign the layering to the element. The other two methods utilize a building block approach to assemble layups and then assigns the layup to a cross section,

which can be from one to a limited number of elements thick. This limit is determined by the parameter MAXIEL. This cross section definition can be calculated for a single representative cross section and then assigned to several similar cross sections, or each cross section can be calculated separately.

Following is a more detailed description of each of the three methods for specifying element layering. In each case, the same example is used to illustrate the method. This example is a composite beam as shown in Figure 25. The beam consists of a symmetric layup of 16 layers with orientation:

0,+45,90,-45,0,+45,90,-45,-45,90,+45,0,-45,90,+45,0

The orientation angles are with respect to the beam axis and are stacked in the global Z direction. All layers are of the same material and thickness. The finite element model for this example beam contains 40 elements as shown in Figure 25.b. The connectivities are such that the structural coordinate system is similar for all elements. A convenient way to check the orientation of the structural coordinate system is shown in Figure 25.c. which labels the element faces.

#### 4.1.4.1. Manual Layering Method

One way to specify the layup is to manually input the layering for the elements in the model, as shown in Figure 26. Note that when using the manual layering method, the option to generate equivalent homogeneous orthotropic material properties is not available.

The layer orientations can be referenced to the element structural coordinate system or directly to the global coordinate system. In this example they are referenced to the elemental system by specifying an angle of rotation about the elemental system axis which is aligned through the thickness of the layers (LAX). Specification of the through the thickness axis also determines the axis from which angles are measured. For LAX=3 zero degrees is measured from the R axis, for LAX=2 zero degrees is measured from the T axis, and for LAX=1 zero degrees is measured from the S axis.

Since there are two elements through the beam thickness, 2 layer definitions must be specified (NELAY=2). Here the upper half (+Z) elements are defined first, then the lower half (-Z)

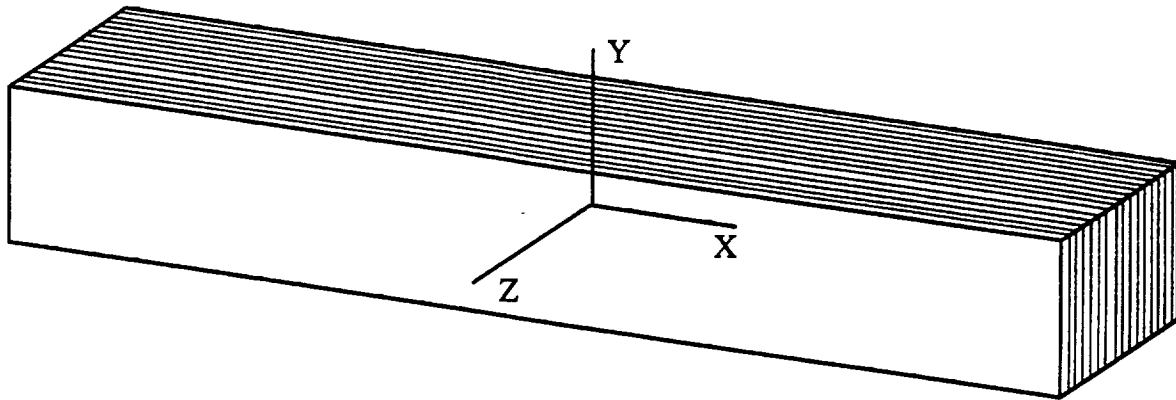


FIGURE 25.a. COMPOSITE BEAM

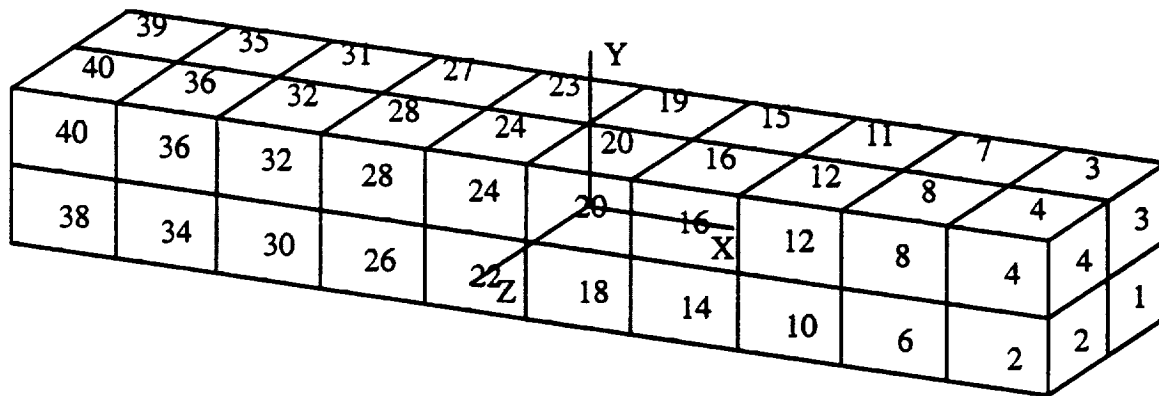


FIGURE 25.b. COMPOSITE BEAM ELEMENT NUMBERING

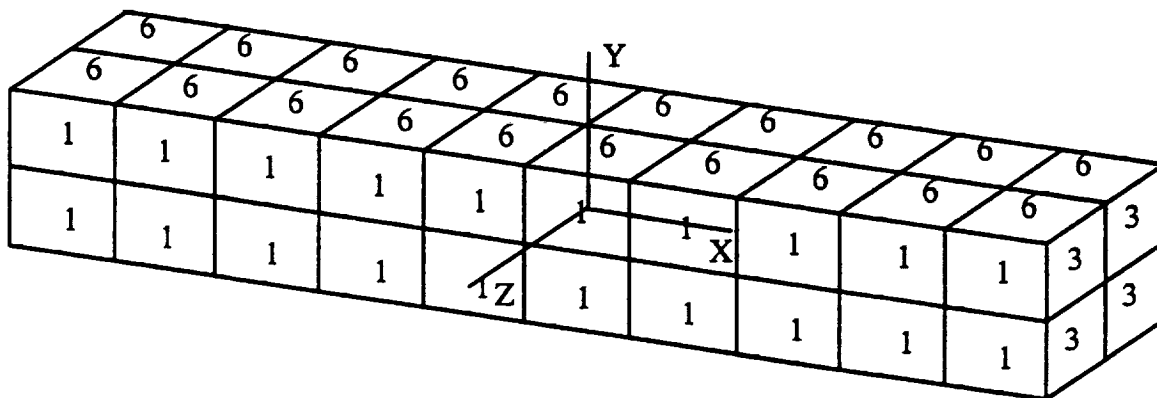


FIGURE 25.c. ELEMENT FACE NUMBERS

2	\$ NELAY (IGAX = 0 since not specified)
2 8 3 0 40 2	\$ NEL NLAYR LAX ISMEAR NEND NINC
1 .125 45.	\$ IMAT PER ANG (LTRN = 0)
1 .125 0.	
1 .125 135.	
1 .125 90.	
1 .125 45.	
1 .125 0.	
1 .125 135.	
1 .125 90.	
1 8 3 0 39 2	\$ NEL NLAYR LAX ISMEAR NEND NINC
1 .125 90.	\$ IMAT PER ANG
1 .125 135.	
1 .125 0.	
1 .125 45.	
1 .125 90.	
1 .125 135.	
1 .125 0.	
1 .125 45.	
0	\$ NGEN

FIGURE 26. MANUAL LAYERING EXAMPLE

elements. Since a layer interface occurs at the element interface in this example, there is no need to split a layer between two elements.

The first set defines the layering for elements 2 through 40 by 2 (i.e. elements 2, 4, 6, 8, ..., 34, 36, 38, 40). The first layer is always at the negative end of the axis input as LAX, which in this case is the element T axis. Therefore, for this case the layers in the first set are input from the beam midsurface, outward. The zero degree axis for this case when LAX = 3 is the element R axis, which is at 90 degrees to the beam axis. This is taken into account in the specification of ANG. Positive rotations are always counter-clockwise as viewed from the positive end of the rotation axis.

The second set defines the layering for elements 1 through 39 by 2, where the layers are input from the beam lower surface to the midsurface. Note that in this method the thicknesses are

input as fractions of the element thickness (PER), and must therefore sum to 1.0 in each definition set. The variable ISMEAR indicates the use of layered elements or homogeneous elements with equivalent bulk material properties.

#### 4.1.4.2. Cross Section Layering With Assignment To Multiple Cross Sections

Another way to specify the layup is to define the layup through the beam cross section and have the program determine the layering for the individual elements. This method makes use of repeating patterns and symmetry in the layup, using what can be thought of as a building block approach. There are three levels in this approach: the individual layer, the repeating set (termed a generation set), and the order in which the repeating sets are applied (called the generation order). These generation orders are applied to cross sections of the model, which are defined by a starting element number and a through-the-thickness or stacking axis. This axis is one of the element (r,s,t) structural coordinate system axes.

In this example case, the layup will be calculated for one input cross section and the resulting element layering will then be assigned to several cross sections. Layered elements will be used as opposed to homogeneous elements with equivalent properties. The input to do this is shown in Figure 27.

```

0          $ NELAY
1          $ NGEN (IGAX= 0)
1 4        $ NSET NLAY
1 .0625 90. $ LMAT THK ANG (LTRN= 0)
1 .0625 135.
1 .0625 0.
1 .0625 45.
1          $ NORDER
1          $ IGEN(1)
1          $ NLAYUP
2 3 1      $ L1 LAX IORDR (LEND= 0)
1          $ NCROSS
1 3 1 0 40 2 $ LMBEG LAX LORDR ISMEAR LXEND LXINC (LMEND=0)

```

FIGURE 27. LAYERING ASSIGNMENT TO MULTIPLE CROSS SECTIONS

The repeating layer patterns are referred to as generation sets. In this case only 1 generation set is needed (NGEN=1). This set is set 1 and contains 4 layers. The cross section is filled from the outer surface, inward so the layers are input in that order. The order in which the generation sets are used to assemble the layups is specified next. In this case only 1 generation order is needed (NORDR=1) and this order is simply to continuously use generation set 1.

The cross section thickness used to calculate the layup is defined next. Only 1 layup will be calculated (NLAYUP=1), and the cross section used begins at the free surface of element 2 and progresses along the T axis using layup generation order 1. Since LEND is 0, the cross section to be filled continues until a free surface is reached. When specifying a cross section between two free surfaces, the beginning element at either end of the layup axis, LAX, can be used. A cross section which does not begin and/or end at a free surface can also be used, in which case the beginning element, L1, is at the negative end of the layup axis and the ending element is indicated by LEND. The layup will be symmetric because LAX is positive. This is done by filling the first half of the cross section thickness according to the specified generation order, then filling the second half with the reverse of the first half. This means that if the midsurface is reached before all the layers in the generation order are used the plies are dropped at the midsurface rather than the outer surface.

Once the cross section layup is defined, it must be assigned to cross sections in the model. The use of layered elements or homogeneous elements for the cross sections must also be defined. This case assigns the single calculated cross section to every cross section in the model and indicates the use of layered elements for all cross sections. Multiple cross sections are specified by indicating the beginning element of the first cross section, the beginning element of the last cross section, and an increment in element number. All elements in each cross section must increment by the same number from the similar element in the previous cross section. The assigned cross sections are assumed to begin and end at free surfaces unless LMEND is not zero, in which case LMBEG must be at the negative LAX end of the cross section and LMEND at the positive LAX end. The variable ISMEAR indicates the use of layered elements or homogeneous elements with equivalent bulk material properties.

The calculation of the layup from the generation order results in layer thicknesses expressed in terms of fractions of the cross section thickness. Therefore, the actual layer thicknesses will be scaled by the thickness of the cross section to which the layup is assigned. The element layering is determined and assigned using the fraction of the total thickness instead of the actual thickness values. This allows the same calculated layup to be assigned to cross sections containing various number of elements. Also, note that LAX does not need to be the same in the cross section being used to determine the total layup and the cross section to which the layup is assigned. However, the layer angles will be referenced from the axis corresponding to LAX of the assigned cross section. In this case since  $LAX = 3$ , zero degrees refers to the R axis.

#### 4.1.4.3. Cross Section Layering With Generation For Each Cross Section

The use of a representative cross section to determine the layup and then assigning this layup to several cross sections as described in Section 4.1.4.2. can not be done if each cross section is of a different thickness. For such a situation the layup is different in each cross section, and thus in each element. The next example uses a method that calculates the layup and assigns the element layering for each cross section. The building block approach is the same, however. Layered elements rather than homogeneous elements with equivalent properties are used once again. The input for this example is shown in Figure 28.

The input for this method is very similar to that shown in Figure 27. with the exception that no layup is calculated before assigning to cross sections. The value of LORDR then refers to the generation order to use instead of the calculated layup.

#### 4.1.4.4. Special Layering Considerations

Some considerations to the layout of the mesh on the structure are necessary to ease the specification of element layering. First of all, it is extremely convenient if the orientation of the structural coordinate system of all elements is basically the same. This means that when looking at a given surface of a model, the face number of elements on that surface are all the same and, more importantly, the through thickness axis will be the same. Also, a consistent element numbering

```

0          $ NELAY
1          $ NGEN (IGAX= 0)
1 4        $ NSET NLAY
1 .0625 90. $ LMAT THK ANG (LTRN= 0)
1 .0625 135.
1 .0625 0.
1 .0625 45.
1          $ NORDER
1          $ IGEN(1)
0          $ NLAYUP
1          $ NCROSS
1 3 1 0 40 2 $ LMBEG LAX LORDR ISMEAR LXEND LXINC (LMEND= 0)

```

FIGURE 28. LAYERING GENERATION FOR EACH CROSS SECTION

pattern is very helpful when assigning or generating layups. This allows the layering of several elements to be specified with a single input line.

The material coordinate system is formed as an orthogonal coordinate system beginning from the structural coordinate system of the element. One axis of the structural coordinate system is used as a basis and remains unchanged in the material coordinate system. This basis axis is crossed into one of the other two structural axes to form the second material system axis. The final material axis is formed as a cross product of the first two. If the through thickness axis is perpendicular to the layer surfaces, it can be used both as the basis for the orthogonal material coordinate system and as the material 3 axis, about which the layers are rotated. If this is not the case, one of the other two element structural coordinate system axes can be specified as the base axis on which to build the orthogonal material coordinate system by designating the through thickness axis, LAX, as a two digit code. The first digit (tens place) is the through thickness axis and the second digit (ones place) is the axis to be used as the basis in forming the material coordinate system.

The material reference (zero degree) axis is determined by the selection of the through thickness axis and is oriented as a result of the selection of the base axis. Depending on the geometry, this can result in different reference orientations in each element. In these cases it may



be desirable to orient the reference axis with respect to the global system while rotating about an axis normal to the layers. This can be done using the IGAX input variable. The value of IGAX indicates the global axis (1=X, 2=Y, 3=Z) and the sign of IGAX indicates whether the reference axis is to be made perpendicular (+) or parallel (–) to the global axis. The global axis indicated by IGAX will be projected onto the material 1–2 plane. The material system will then be rotated about the through thickness axis until the reference axis is perpendicular ( $IGAX > 0$ ) or parallel ( $IGAX < 0$ ) to this projected global axis.

To illustrate the meaning of the previous paragraphs refer to the elements in Figures 28.a., 28.c., and 28.b. These figures show three elements layered such that the layers lie in the X–Z plane. The exposed face numbers are printed on the figures to show the element structural coordinate system.

The element in Figure 28.a. is a cube so that the structural coordinate system is exactly aligned with the global system throughout the element. The S axis is the through thickness axis and it is parallel to the global Y axis as indicated by the face numbers. In this case LAX=2 can be specified, which will result in a material system parallel to the global system after performing the cross product calculations. The reference axis in this case is the structural T axis which lies parallel to the global Z axis, and the 90 degree axis is the structural R axis which is parallel to the global X axis.

The element in Figure 28.b. is skewed such that the S axis is not perpendicular to the R–T plane, in general. The R axis is always parallel to the global X axis and the T axis is always parallel to the global Z axis because each X–Z cross section remains rectangular. In this case LAX=21 or LAX=23 could be specified, either of which will result in a material system parallel to the global system with the reference axis along global Z.

The element in Figure 28.c. is skewed even more so that neither the S axis or T axis is consistent throughout the element. The R axis remains parallel to the global X, however, because the sides of the element remain parallel to the X axis. In this case LAX=21 is the only choice that would form a material system aligned with global. Use of LAX=23 would result in a system with the material 3 axis perpendicular to the layers, but the reference axis would not generally be

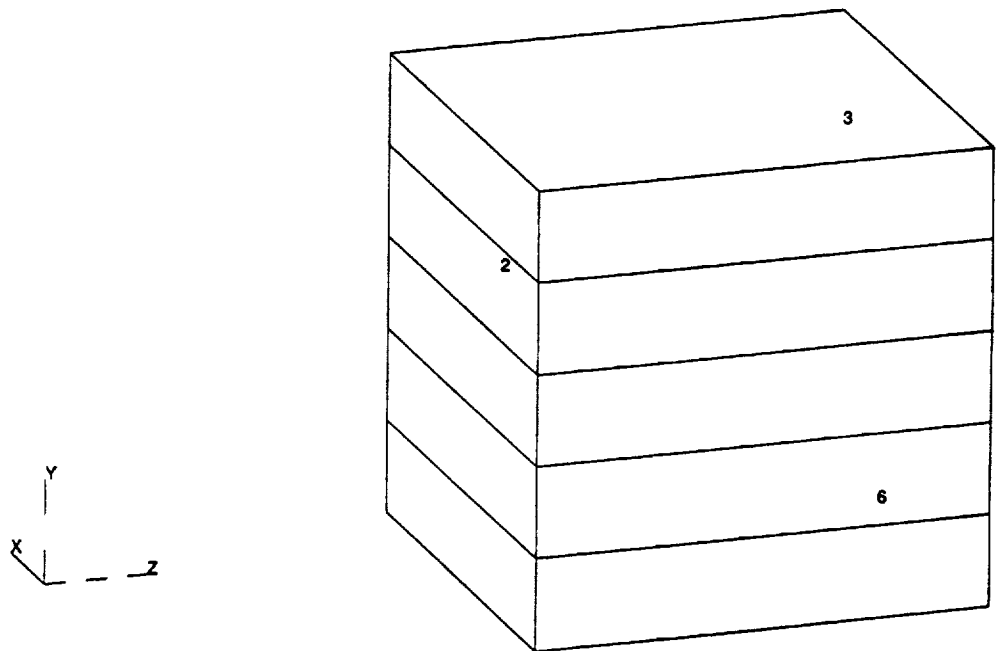


FIGURE 28.a. ORTHOGONAL STRUCTURAL COORDINATE SYSTEM

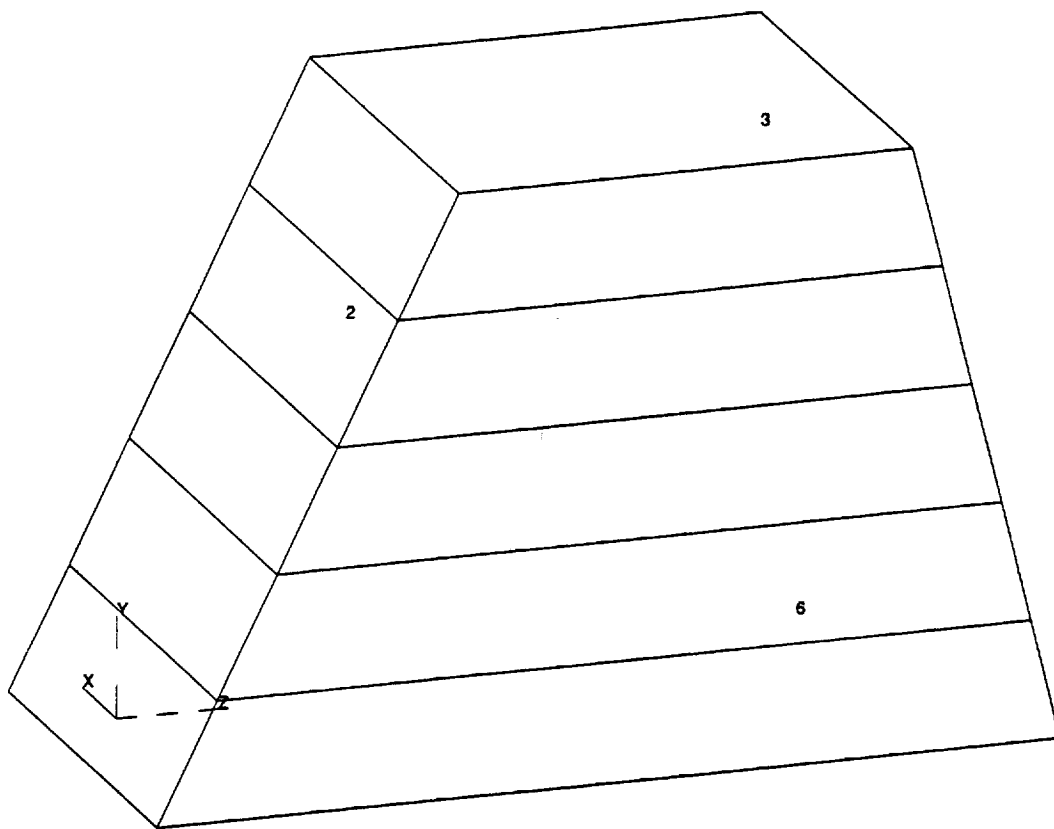


FIGURE 28.b. NON-ORTHOGONAL THROUGH THICKNESS AXIS

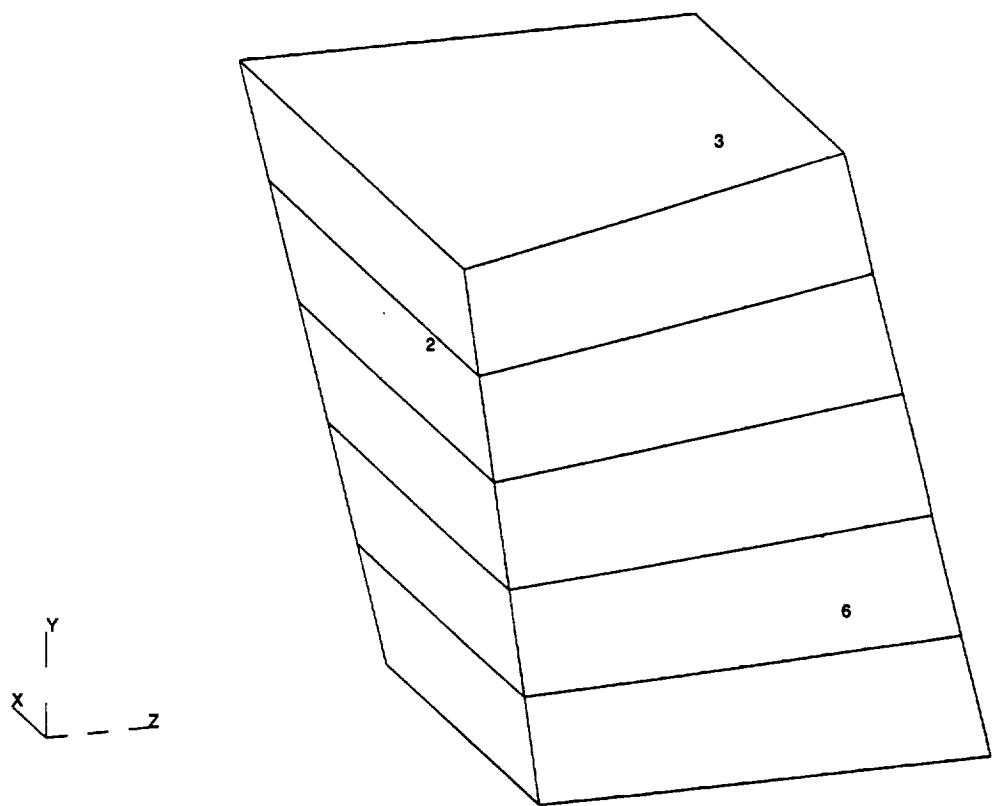


FIGURE 28.c. R AXIS CAN BE USED AS MATERIAL BASIS AXIS

aligned with the global Z axis. If IGAX=1 was specified together with LAX=23, the resulting material system would be aligned with global. Specifying IGAX=1 will result in a rotation of the material system about the material 3 (through thickness) axis until the reference axis is perpendicular to the global X. Because the material 3 axis is aligned with the global Y axis, the reference axis will be aligned with the global Z (perpendicular to both Y and X). If the material 3 axis made an angle with the global Y (as would be the case if LAX=2 was specified) and IGAX=1 was used the resulting reference axis would lie in the global Y-Z plane at an angle to the Z axis that would be the same as that between the material 3 axis and global Y.

For models in which the layup is radial-tangential in nature, the IGAX variable can be used to indicate the material 1 reference axis is to be oriented with respect to a tangential axis. The radial-tangential axis system is aligned with the global system at zero tangential rotation. IGAX is entered as a two digit combination, where the tens digit indicates which global axis is aligned with radial and the ones digit indicates which global axis is aligned with tangential. The coordinates of the integration point are used to calculate the angle of the point in the radial-tangential system. The material axis system is then rotated about the material 3 axis until the material 1 reference axis is perpendicular ( $IGAX > 0$ ) or parallel ( $IGAX < 0$ ) to the projection of the tangential axis on the material 1-2 plane.

The stiffness integration points for layered elements are located at the layer midplanes, thus the distribution of layers in the element thickness direction determines the location of these integration points. A Gauss distribution of the integration points gives a good integration of stiffness with a minimum of points. Since layering generally moves these integration points from this optimal location, more than the minimum number through the thickness will generally be required to integrate the element stiffness as accurately. This means that there should be at least three layers in an element, and probably better to have five or more (see Figure 8.). Also, layers that have roughly the same thickness in an element give the best integration of the element stiffness. Elements composed of a very thick layer or two and then some thin layers may not have their stiffnesses integrated as accurately as if the thick layer was broken down into several layers of

thickness on the order of magnitude of the thin layers. It should be kept in mind though, that the more layers in an element, the longer it will take to calculate the stiffness.

#### 4.1.4.5. Layer Orientation Visualization

To aid in visualizing the material orientation resulting from the layering input, a printout can be obtained which contains the direction cosines of the material reference axis scaled by the magnitude of the Jacobian for this axis. This is printed at one integration point for each layer of each element. The particular integration point is indicated by the input variable LYPRNT, which is the flag controlling whether the layer visualization file is to be printed (see Section V.1. of the input sheets). This can be used with a two node beam element with one end positioned at the element centroid and the other end displaced from the centroid by the scaled cosines. The format of this file is:

Layer No., Node No., Dx, Dy, Dz, NNMAX

The node number is based on the element in which the layer occurs according to the equation  $2*(LX-1)+NNMAX+1$ , where LX is the element number and NNMAX is the maximum number of nodes allowed. The scaled direction cosines are Dx, Dy, and Dz. The layers in an element are numbered from 1 to the number of layers in the element with layer 1 at the negative end of the LAX axis. For models with more than one element through the thickness there will be more than one layer 1 in each cross section.

#### 4.1.5. CONSTRAINT EQUATIONS

Constraint equations enforce a relationship between nodal degrees of freedom. This relationship and the degrees of freedom that they relate are defined by the user input. The form of the constraint equation input as used in CSTEM is described in Section IV. of the input sheets. Essentially, the constraint equation defines a dependant degree of freedom as some linear combination of various independant degrees of freedom. There is a limit to the number of independant degrees of freedom. Currently MXCST is set to 20. There is also a limit to the total number of degrees of freedom which may be related to a single element. This includes the normal

degrees of freedom for the element as well as any additional degrees of freedom that may be included due to the constraint equations. This limit, MDOFE, is currently set to 90.

#### 4.1.6. ADDITIONAL STIFFNESSES

Additional stiffnesses are stiffness terms relating the translation and rotation DOF between a pair of nodes. Additional stiffnesses are described in Section X. of the input sheets. These may be used for many purposes and are often used as spring constants to apply soft ground springs to a structure in order to perform a free-free vibration analysis. Lumped mass and rotary inertia terms can also be input for the same DOF as the additional stiffness terms. This allows simulation of system effects on a model. Stiffnesses and masses are entered in terms of global DOF.

#### 4.1.7. TRANSFORMATIONS

Transformations are used in many places throughout the CSTEM code such as for rotated nodal boundary conditions, orthotropic material transformations, and electromagnetic loading. There are three ways to specify transformations. In each method the transformations are associated with a unique transformation number by which the transformation will be identified in other parts of the input deck. The input transformation matrix should transform from global to local. See Section 2.3. for a more detailed explanation on how transformations are used, while Section VIII. of the input sheets pertains to the specification of transformations.

The first method for specifying a transformation uses three reference nodes to define a base axis and plane. The first node forms an origin, the second a local X axis, and the third forms the local X-Y plane and indicates the direction of the positive Y axis. The reference nodes may be structural nodes, but extra non-structural nodes (which are not subject to the node limits) may be entered immediately prior to specification of the transformations. The second way to specify transformations is to enter the direction cosines of the transformation matrix directly. Only the direction cosines of the local X and Y axes are input, while the local Z axis is determined by crossing local X into local Y. The third way to specify transformations is to use Euler angles. Beginning with the global system, the rotations about the global X axis, the resulting local Y axis

and the final local Z axis are entered so that the global system will be rotated to align with the desired local system.

#### 4.1.8. NODAL FIXITIES

Fixed nodal degrees of freedom are treated differently than non-zero prescribed nodal displacements. When a degree of freedom is specified as fixed, the equation associated with that degree of freedom is not included in the global set of equations. This means that the degree of freedom cannot be changed to a non-zero value at any time during the analysis. Section IX. of the input sheets pertains to nodal fixities. The fixed node number is followed by six indicators corresponding to six degrees of freedom. These six DOF are the translational degrees of freedom along the X, Y, and Z axes and the rotational degrees of freedom about the X, Y, and Z axes respectively. These DOF are in the global system unless a transformation is specified in Section III. for the fixed node. A fixed degree of freedom is indicated by a 1 in the appropriate location. The 3D elements used in CSTEM have three translational degrees of freedom only at each node, so the rotational degrees of freedom are automatically fixed regardless of the input value.

#### 4.1.9. NODAL DISPLACEMENTS AND FORCES

Nodal forces and prescribed displacements are applied in the three directions of the active coordinate system for the node. This active system is the global system unless otherwise specified by assignment of a transformation to the node. Sections XIX. and XX. pertain to nodal prescribed displacements and forces respectively. These values can be changed from load case to load case. The degree of freedom in which the boundary condition is applied is indicated as an integer 1, 2, or 3 corresponding to the translational DOF along the X, Y, or Z axis.

#### 4.1.10. NODAL TEMPERATURES

Nodal temperatures can be input for each load case, or calculated by running a coupled heat transfer analysis. Section XXI. of the input sheets pertains to specifying nodal temperatures by input. When doing a heat transfer analysis the initial temperatures are input in this way. Nodal



temperature input ignores specification of node numbers which do not exist. This is done to ease input of a temperature for all nodes when the node numbers are not consecutive and inclusive.

#### 4.1.11. PRESSURE LOADS

Pressures are loads applied to element faces. The pressure loads are converted internally to the nodal loads necessary to simulate the pressure, consistent with the order of the element itself. These loads may vary across the face and may be changed from load case to load case. Section XXII. of the input sheets pertain to pressures. Positive pressure induces compression in the element. An element may have pressure loads on all six faces at once. The variation in pressure across an element face is indicated by specifying the pressure value at each of the nodes comprising the face. If only one pressure value is entered it is assumed that the entire face is subjected to a constant pressure of this value.

For large displacement problems, the geometry is updated from load case to load case. The pressure normally is assumed to remain perpendicular to the deformed element face and nodal loads are calculated using the face geometry at the beginning of each equilibrium iteration. By entering the face number as a negative number, the undeformed face configuration can be used when calculating the nodal loads.

#### 4.1.12. PLASTICITY ANALYSIS

Plasticity is time independent nonlinear material behavior. A more detailed description of the theory behind the plasticity models implemented in CSTEM can be found in Section 3.1. 11. 2. while this section describes the actual plasticity analysis input variables. Section XV. of the input sheets pertains to plasticity analysis input. Plasticity models are activated as a function of material type, so that more than one plasticity model may be active within a single analysis.

##### 4.1.12.1. Activating A Plasticity Analysis

The variable IPLO, described in Section 4.1.1.1.4., must be set to a nonzero value in order for a plasticity analysis to be performed. IPLO also defines the iteration method to be used. A load case control variable, IPC, controls whether the plasticity input variables described in this section

are read in a load case. Plasticity analysis input will be read when  $IPC=1$ , as described in Section 4.1.1.2.1.. Plasticity analysis input must be read before a plasticity analysis can be performed. Once the plasticity parameters and properties have been read it is not necessary to input them in every subsequent load case. Only if the user wants to change any of the plasticity parameters or properties from previously input values do they need to be entered in a subsequent load case.

One of the plasticity parameters read when  $IPC=1$  is the plasticity model selection flag,  $IPLAS$ . This variable can be used as a local turn off switch for plasticity analysis by setting  $IPLAS = -1$ .  $IPLAS$  can then be reset and plasticity analysis continued in a subsequent load case.  $IPLAS$  is initially set to  $-1$  until the user selects the desired model, activating the plasticity analysis. There are currently four different plasticity models from which the user may choose, including a user defined model. There are two isotropic material models: simple isotropic hardening ( $IPLAS=0$ ), which is not intended for use with fully reversed plasticity cyclic loadings, and a subvolume model ( $IPLAS=1$ ), which is a kinematic hardening model by default, but has some hardening adjustment capabilities. There is one orthotropic plasticity model ( $IPLAS=3$ ) developed for use with fibrous composite materials. The user plasticity model option ( $IPLAS=10$ ) requires the user defined plasticity model to be compiled as part of the CSTEM code.

#### 4.1.12.2. Plasticity Iteration Control

Plasticity analysis is an iterative procedure. The maximum number of iterations allowed in a load step is specified by the variable  $MPIT$ . This parameter is independent of material type. If convergence is not achieved in  $MPIT$  iterations, an unconverged solution is declared. It is recommended to stop execution if an unconverged solution is obtained, however by specifying  $MPIT$  as a negative number, the user can be daring and continue execution from the unconverged solution.

Another parameter which controls plasticity iterations is the convergence tolerance,  $PTOL$ . This is a material dependent parameter, since different values can be specified for different materials. Convergence is determined by comparing the change in plastic strain increment between subsequent iterations to the convergence tolerance. If the change is less than the convergence

tolerance at every integration point, convergence is achieved. If PTOL is entered as a negative number, a percent change is used. A default tolerance of 1.E-5 is used if zero is entered for PTOL.

#### 4.1.12.3. Plasticity Material Properties

The material properties required for plasticity vary somewhat according to the plasticity model being used. In general, curves of stress vs. total strain at a single or multiple temperatures for each material are required. A detailed explanation of required input follows by plasticity model type.

##### 4.1.12.3.1. Isotropic Material Plasticity

Isotropic plasticity input is described in Section XV.3. of the input sheets. Both the simple isotropic hardening plasticity model (IPLAS=0) and the subvolume kinematic hardening plasticity model (IPLAS=1) take the same of material inputs. This consists of stress-strain pairs which form a piecewise continuous uniaxial stress vs. total strain curve. A curve must be entered for each material defined elastically and for the same temperatures as the elastic properties of that material. The temperatures themselves are not entered as part of the plasticity input, the stress-strain curves are simply assigned to the elastic property temperatures according to the order of input.

An initial line of input is entered which defines the material number (IMAT), the number of stress-strain pairs in each curve (NPS) for this material, and the convergence tolerance (PTOL) for this material. If NPS is zero or negative, no stress-strain pairs will be read.

This initial input line is followed by the uniaxial stress-strain curve data, which is entered a curve at a time as stress and total strain pairs, one pair per line, stress in psi then total strain in inches/inch. The first stress-strain pair in each curve defines the beginning of the nonlinear portion of the curve and should be consistent with the elastic modulus at that temperature. There are NPS stress-strain pairs in each curve and the curves are entered in the order corresponding to the temperatures entered for the elastic constants of the same material.

#### 4.1.12.3.2. C.T. Sun Orthotropic Plasticity

Orthotropic plasticity input is described in Section XV.4. of the input sheets. The orthotropic plasticity model (IPLAS=3) requires the definition of potential function parameters in addition to the master effective stress vs. effective plastic strain curves. The potential function parameters should be determined such that the master stress-strain curves are independent of the in plane orientation of the orthotropic material. Definition of the potential function parameters allows calculation of effective stress from the uniaxial stress test component (equation (96)) and then calculation of effective plastic strain (equation (99)). The master effective stress vs. effective plastic strain curve can be entered as stress-strain pairs defining a piecewise continuous curve or as coefficients defining a power law representation of the curve. Orthotropic plasticity inputs must be entered for each material defined elastically, however the number of temperatures and the temperature values need not be the same as those used to define the elastic constants. The input range of temperatures should bracket the temperatures to be encountered in the analysis.

An initial line of input defines the material number (IMAT), the number of temperatures (NTEM) at which properties will be defined for this material, the number of stress-strain pairs (NPS) in each master stress-strain curve for this material, and the convergence tolerance (PTOL) for this material. If NPS is entered as a zero, a power law representation of the master stress-strain curve will be read. A positive NPS value will cause NPS stress-strain pairs to be read. A negative value for NPS will cause no master stress-strain curve parameters to be read. This may be useful when performing orthotropic creep and no plasticity.

For each of the NTEM temperatures, the potential function parameters are entered followed by the master stress-strain curve for that temperature. The inputs should be entered in either increasing or decreasing temperature order. The temperature value is input on the same line as the potential function parameters  $A_{12}$ ,  $A_{23}$ ,  $A_{13}$ ,  $a_{44}$ ,  $a_{55}$ , and  $a_{66}$ . The parameters  $a_{12}$ ,  $a_{23}$ , and  $a_{31}$  can be calculated from elastic constants and not input or may be entered on this same line of input. After entering the potential function parameters for the temperature, the master stress-strain curve is entered either as NPS effective stress, effective plastic strain pairs, one pair to a line, or as three constants ( $A$ ,  $n$ , and  $C$ ) defining the master effective stress vs. effective plastic strain curve as a

power law function,  $\bar{\epsilon}^p = A\bar{\sigma}^n + C\bar{\sigma}$ . The potential function parameters are unitless, while the master stress-strain curves are for stress in psi and strain in inches/inch.

#### 4.1.12.3.3. User Plasticity

User plasticity is described in Section 3.1. 11. 2.4. User plasticity (IPLAS=10) requires the users plasticity model code be compiled with the CSTEM code. Access to the CSTEM iteration procedure is through the subroutine CHOOK. User plasticity material properties and model parameters can be entered, stored and retrieved using the UMATIO subroutine. The input required by CSTEM are the lengths of the CHOOK and UMATIO vectors, and the UMATIO vectors themselves.

The CHOOK vector lengths, NIPS, NRPS, NEPS, NSTATP, and NSTATM, must include the reserved, predefined values. If the user has no data to store in these vectors, the minimum value that should be input for each of them is NIPS = 10, LRPS = 15, LEPS = 18, LSTP = 30, and LSTM = 0. Any additional data that the user plasticity model needs to store in these vectors will increase the vector lengths from these minimum values. The full vector lengths are input.

The UMATIO vectors have no predefined length, with the exception of use with the damage model. When the damage model is not activated, the value of MQMX is the maximum length needed for any one material specification. The user plasticity material properties need not all have the same number of items for each material, but MQMX should indicate the largest number (maximum vector length) that a material will need.

After specifying the vector lengths, two lines of input containing plasticity and creep iteration parameters are needed. Both lines are input whether or not both user models are used. The first line contains the maximum number of plasticity iterations allowed (PLITER) and a convergence tolerance (PTOL) which may be used at the users discretion. Note that if the user does not check for convergence, CSTEM checks for convergence using a default convergence tolerance and technique (1.E-5 max difference in Von Mises effective plastic strain increment between iterations, checked at each integration point). Creep iteration parameters are similar to the plasticity iteration parameters.

User material vectors must be entered for each material defined elastically. A separate line defining the material number (MTN) and length of the user material vector (MATQ) is entered. This is followed by the material properties and other material parameters that a user plasticity model may require. These properties and parameters may be entered on any number of lines and with any number of inputs per line. The inputs will be read until MATQ quantities have been read and stored into the material vector for material MTN. These properties will be made available to the user model through the UMATIO routine in exactly the same order as read from the input deck. Any dependencies, such as temperature dependence, can be contained in the user model material vectors at the discretion of the user, who will have to interpolate or utilize this data as required.

#### 4.1.13. CREEP ANALYSIS

Creep is time dependent nonlinear material behavior. A more detailed description of the theory behind the creep models implemented in CSTEM can be found in Section 3.1. 11. 4. while this section describes the actual creep analysis input variables. Sections XVI. and XVII. of the input sheets pertain to creep analysis input. Creep models are activated as a function of material type, so that more than one creep model may be active within a single analysis.

##### 4.1.13.1. Activating A Creep Analysis

The variable ICRO, described in Section 4.1.1.1.4., must be set to a nonzero value in order for a creep analysis to be performed. Setting ICRO to a nonzero value will cause the time increment controls to be read. Time increment control variables are entered for every load case. The size of the time step for the load case is defined by the variable, TIME, entered on the load case control line. The time step should be greater than zero to perform creep analysis during the load case. No creep will occur for a zero time step.

Another load case control variable, IPC, controls whether the creep analysis parameters material input variables are read in a load case. Creep analysis input will be read when IPC=2, as described in Section 4.1.1.2.1.. Creep analysis input must be read before a creep analysis can be performed. Once the creep analysis parameters and properties have been read it is not necessary to input them in every subsequent load case. Only if the user wants to change any of the creep

parameters or properties from previously input values do they need to be entered in a subsequent load case.

One of the creep parameters read when IPC=2 is the creep model selection flag, ICRP. This variable can be used as a local turn off switch for creep analysis by setting ICRP = 0. ICRP is initially set to 0 until the user selects the desired model, activating the creep analysis. There are three creep model options, including a user creep model. The creep model used is selected by the variable ICRP, where ICRP=1 for isotropic material creep, ICRP=2 for orthotropic material creep, and ICRP=10 for a user creep model. The type of creep hardening is specified by the variable IHARD, where IHARD=0 indicates time hardening and IHARD=1 indicates strain hardening.

#### 4.1.13.2. Time Increment Control

Time incrementing is described in more detail in Section 3.1. 11. 4.1. The time increment control variables are all entered on a single line in each load case of a creep analysis. For the simple division of a time step into equal increments, only the variable N2M needs to be entered to define the number of increments. To use dynamic time incrementing, N2M is entered as 0 and other variables on the line may need to be input, although default values will be used if no other values are given.

The initial time increment size for the load case (TINIT) should be chosen small enough so that the creep analysis converges in this first time increment, but not so small that an unnecessary number of increments will be taken due to the limit on the increase of time increment size. The default for the first load case is 1% of the time step. For subsequent load cases the default is half of the final time increment used in the previous load case.

The remaining inputs on this line define the time increment size with the maximum effective creep strain increment allowable in a time increment (ECMAX), the maximum effective stress increment allowable in a time increment (SIGMAX), the maximum allowable integration error in a time increment (ERMAX), the minimum allowable time increment size (DELMIN), and the maximum multiplier allowed in going from one time increment to the next (DELMUL). The use of these inputs may best be explained by referring to the equations of Section 3.1. 11. 4.1.,

where  $SIGMAX = \left( \Delta \sigma_{eff} \right)_{allowable}$  in equation (101),  $ECMAX = \left( \Delta \epsilon_{eff}^c \right)_{allowable}$  in equations (102) and (105), and  $ERMAX$  is the variable *tol* used in equation (105). The defaults for these variables are listed in the input sheets, Section XVI.

#### 4.1.13.3. Creep Iteration Control

Creep analysis is an iterative procedure. The maximum number of iterations allowed in a time increment is specified by the variable *MCIT*. This parameter is independent of material type. If convergence is not achieved in *MCIT* iterations, an unconverged solution is declared. It is recommended that execution be stopped if an unconverged solution is obtained, however by specifying *MCIT* as a negative number, the user can continue execution from the unconverged solution.

Other controls on creep iteration are material dependent, but are identical for the two creep models implemented in CSTEM. Convergence can be obtained as either an absolute difference in the Von Mises effective stress from one iteration to the next or as a percent difference in this stress. The variable *ICRC* indicates which method to use, 0 for stress difference, 1 for percent difference. The convergence tolerance, *CTOL*, is entered as psi or decimal percent (.01 for 1%) according to the value used for *ICRC*. A temperature cutoff, *TCUT*, defines the temperature below which no creep will occur. Although extrapolation of creep constants outside the range of input temperatures is allowed, it is good practice to specify the temperature cutoff as the lowest temperature at which creep constants are entered to avoid this extrapolation. (Of course this requires that creep constants be entered over the actual temperature range in which any significant creep occurs.)

#### 4.1.13.4. Creep Material Properties

The material properties required for creep vary somewhat according to the creep model being used. In general, creep equation constants at a single or multiple temperatures for each material are required. A detailed explanation of required input follows by creep model type.



#### 4.1.13.4.1. Isotropic Material Creep

The material number for which the input applies (IDM) is listed on the same line as the cutoff temperature (TCUT), creep convergence parameters (ICRC and CTOL), and normalization stress. The normalization stress is in psi. It is the value divided into the effective stress used in the creep equation, since the effective stress used in the creep model is in psi. This allows creep constants for stress in ksi or some other units to be used if desired. Note that the normalization stress defaults to 100000 if no value or zero is entered.

Creep constants can be entered at up to ten temperatures. These temperatures need not correspond to the temperatures at which elastic properties are defined or to temperatures at which creep properties for other materials are defined. The temperatures (CTEM) are all entered on a single input line in increasing or decreasing order. If a single temperature is entered, the creep constants will be the same at all temperatures. Following the temperatures, the creep constants are entered in the order corresponding to the temperature input. There are five constants for this model,  $K$ ,  $n$ ,  $m$ ,  $Q$ , and  $r$ , describing the creep equation (equation (106), repeated here)

$$\epsilon_{eff}^c = K \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^n t^m + Q \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^r t \quad (106)$$

The units for constants  $K$  and  $Q$  should be consistent with the choice of normalization stress.

#### 4.1.13.4.2. C.T. Sun Orthotropic Creep

The material number for which the input applies (IDM) is listed on the same line as the cutoff temperature (TCUT), creep convergence parameters (ICRC and CTOL), and normalization stress. The normalization stress is in psi. It is the value divided into the effective stress used in the creep equation, since the effective stress used in the creep model is in psi. This allows creep constants for stress in ksi or some other units to be used if desired. Note that the normalization stress defaults to 1 if no value or zero is entered.

Creep constants can be entered at up to ten temperatures. These temperatures need not correspond to the temperatures at which elastic properties are defined or to temperatures at which creep properties for other materials are defined. The temperatures (CTEM) are all entered on a

single input line in increasing or decreasing order. If a single temperature is entered, the creep constants will be the same at all temperatures. Following the temperatures, the creep constants are entered in the order corresponding to the temperature input. There are seven creep constants,  $B$ ,  $C$ ,  $D$ ,  $m$ ,  $n$ ,  $r$ , and  $s$  describing the creep equation (equation (107), repeated here)

$$\epsilon_{eff}^c = C \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^n \left[ 1 - \exp \left[ - D \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^r t^s \right] \right] + B \left( \frac{\sigma_{eff}}{\sigma_{norm}} \right)^m t \quad (107)$$

The units for constants  $C$ ,  $D$ , and  $B$  should be consistent with the choice of normalization stress.

In addition to the creep constants, the C.T. Sun potential function parameters must be input as well. The temperatures at which potential function parameters are entered need not correspond to the creep constant temperatures, although both should bracket the expected temperature range of the analysis. The potential function parameters are entered as part of the plasticity input described in Section 4.1.12.3.2. and in the input sheets, Section XV.4.a. Usually a plasticity analysis will be performed at the same time as a creep analysis. If this is not the case, the potential function parameters must still be read. This can be accomplished by setting the global plasticity flag,  $IPLO=0$  to indicate no plasticity analysis, then setting the load case control flag,  $IPC=12$  to read both plasticity and creep properties and  $IPLAS=3$  to indicate CT Sun plasticity. The potential function parameters will be read as well as the effective stress-strain curve. Dummy values may be entered for the stress-strain curve, or a negative value for NPS may be specified which will avoid reading the stress-strain curves.

#### 4.1.14. DAMAGE MECHANICS ANALYSIS

This is an evolving portion of the CSTEM code. Damage mechanics analyses calculate inelastic damage strains as well as material damage which will change the elastic properties of the material. Damage models are activated as a function of material type, so that more than one damage model may be active within a single analysis or damage may be activated for only a certain set of specific materials. There is currently one damage mechanics model, which is for CMC composite materials. It continues to be developed by Fred Leckie, et.al. at UCSB. There is also a

simple polynomial stiffness loss model as well as a method to include a user defined damage mechanics routine using the DHOOK subroutine.

#### 4.1.14.1. Activating A Damage Analysis

The variable IDMO, described in Section 4.1.1.1.4., must be set to a nonzero value in order for a damage mechanics analysis to be performed. IDMO also defines the iteration method to be used. A load case control variable, IPC, controls whether the damage mechanics input variables described in this section are read in a load case. Damage mechanics analysis input will be read when IPC=3, as described in Section 4.1.1.2.1. Damage mechanics analysis input must be read before a damage mechanics analysis can be performed. Once the damage mechanics parameters and properties have been read it is not necessary to input them in every subsequent load case. Only if the user wants to change any of the damage mechanics parameters or properties from previously input values do they need to be entered in a subsequent load case.

The model used in a damage analysis is selected with the variable MDAMG. This variable can also be used as a local turn off switch for damage analysis by setting MDAMG = 0. Damage mechanics can be continued in a subsequent load case by setting MDAMG to the desired value. MDAMG is initially set to 0 for every material until the user selects the desired model, activating the damage analysis. There is currently one damage mechanics model, which is for CMC composite materials. This model is activated by setting MDAMG=2. A simple polynomial stiffness degradation model is available by setting MDAMG=3. The user damage model option (MDAMG=10) requires the user defined damage model to be compiled as part of the CSTEM code.

#### 4.1.14.2. Damage Analysis Iteration Control

Damage mechanics analysis is an iterative procedure. The maximum number of iterations allowed for convergence to be achieved is set by the variable MDIT. If there is no convergence in MDIT iterations, the iterative process is stopped and there is no convergence. It is recommended

that execution stop if an unconverged solution is obtained, however by specifying MDIT as a negative number, the user can continue execution from the unconverged solution.

Cyclic loadings can be simulated in one of two ways, or as a combination of both. Single or multiple load cases can be run repeated times to explicitly simulate cyclic loading using the mission cycling capability as specified using the input variables NCYC and NCYPTS. This is in addition to the fatigue cycles input to the damage mechanics model using the input variable NCYCM. Note that currently, only the CMCUMAT damage mechanics model has any cyclic fatigue damage capability. A typical static load case is indicated by setting all three cyclic variables (NCYC, NCPYTS, and NCYCM) to 1. This is the default if these variables are left unspecified.

The mission cycling capability allows the analysis of more complex cycles than can be calculated strictly within CMCUMAT. The CMCUMAT fatigue calculations can be utilized to degrade the material properties due to a large number of minor cycles (using the NCYCM input variable), followed by an occasional excursion to a different loading condition simulated with the mission cycling capability. The time independent static capability of CMCUMAT is used with mission cycling, so that the degradation in material properties due to the fatigue cycling calculation within CMCUMAT would contribute to a different response and potential further material degradation in the mission cycle.

The mission cycle may be composed of multiple load cases. The load cases are input in the usual manner, with the NCYPTS input parameter indicating the number of load cases contributing to the mission cycle. The NCYPTS input is entered in the load case defining the first point of the cycle. The NCYC input variable defines how many times the load cases defining all the points of the cycle are to be run. An analysis which does not use the mission cycling capability would specify NCYC=1, NCYPTS=1, and NCYCM= the desired number of cycles to be calculated by the damage mechanics model.

The number of cycles to be calculated by the damage mechanics model is specified using the variable NCYCM. This variable defaults to a value of 1 if any number less than 1 is entered. One cycle indicates that the load case represents the usual change in loading to the values input for the load case. If some value greater than 1 is entered, the damage analysis will be performed

NCYCM number of times for the input load levels. This allows a single load case specification to calculate damage over multiple cycles when used with a damage mechanics model which can account for damage as a function of cycling at a given load level. Refer to Section XVIII.1. of the input sheets to determine the specific input format for the mission cycling variables NCYC, NCYPTS, and NCYCM.

Due to a units conversion required internally, the user must specify whether the damage inputs are supplied in the usual British units (psi, inch) or metric units (MPa, meter). This is done with the IUNIT variable, where 0 indicates British units and 1 indicates metric. If IUNIT is left blank, it is interpreted to be 0 per the usual input convention.

There are two material dependent damage iteration controls. Convergence is determined by comparing the change of some parameter from one iteration to the next. If this change is less than some tolerance at every integration point, convergence is achieved. The convergence type flag, ITYP, indicates the parameter to be compared in subsequent iterations. For ITYP=1, convergence is to be based on the change in a scalar damage parameter. The actual damage parameter may vary from one damage model to another. For ITYP=2 convergence is to be based on the change in effective total strain. Setting ITYP=-10 will cause damage to be calculated without iterating. This assumes no redistribution of stress due to damage during the damage calculations. The variable DTOL indicates the tolerance value. For a positive value of DTOL, convergence is based on the absolute change in the convergence parameter. For a negative value of DTOL, convergence is based on the percent change in the convergence parameter.

#### 4.1.14.3. CMCUMAT Damage Mechanics

The CMCUMAT damage mechanics model is activated by setting MDAMG=2. The material inputs for this model are shown in Figure 29. as the current state vector utilized by the CMCUMAT model. The input sequence is further described in Section XVIII.3. of the input sheets. Figure 29. shows that there are locations available in the state vector for enhancement and expansion of the model. These material inputs are entered on 6 lines of input, 8 items per line at up

to ten temperatures. These temperatures need not correspond to the temperatures at which elastic properties or other material model properties are defined.

#### 4.1.14.4. Polynomial Stiffness Degradation Model

The polynomial stiffness degradation model can be used as a simple method to change the in plane elastic moduli ( $E_1$ ,  $E_2$ ) as a function of stress and cycles. This change is calculated according to equation (100), which is repeated here:

$$\frac{E}{E_0} = 1 + \{A \ln(N) + B \ln(N^2) + C \ln(N^3) + D \ln(N^4)\} * \left\{ S_A + S_B \left( \frac{\sigma}{S_U} \right) + S_C \left( \frac{\sigma}{S_U} \right)^2 \right\} \quad (100)$$

The constants  $A$ ,  $B$ ,  $C$ ,  $D$ ,  $S_A$ ,  $S_B$ ,  $S_C$ , and  $S_U$  are input as a function of temperature.

- PROPS(1–8)            Matrix Elastic Properties
- (1) Young's modulus of Matrix,  $E_m$  [psi]
  - (2) Poisson's ratio of Matrix,  $\nu_m$
  - (3) Volume fraction of Matrix,  $f_m$
  - (4) Weibull exponent for Matrix,  $m_m$
  - (5) Energy release rate density,  $Y_m$  [psi]
  - (6) Saturation value of matrix damage,  $D_{sat}$
  - (7–8)                unused
- PROPS(9–16)        Fiber Elastic Properties
- (9) Young's modulus of Fiber,  $E_f$  [psi]
  - (10) Poisson's ratio of Fiber,  $\nu_f$
  - (11) Volume fraction of Fiber,  $f_f$
  - (12) Weibull exponent for Fiber,  $m_f$
  - (13) Interface shear resistance for Fiber,  $\tau_{th0}$  [psi]
  - (14) Reference length for Fiber,  $L_0$  [in]
  - (15) Radius of Fiber,  $R$  [in]
  - (16) Reference stress for Fiber,  $S_0$  [psi]
- PROPS(17–24)       Composite Properties
- (17) Orientation of Composite,  $\gamma$  [degrees] (set internally)
  - (18–24)            unused
- PROPS(25–32)       Inelastic properties of Matrix (normal stresses)
- (25) Slope of inelastic strain,  $a_0$  [1/psi<sup>2</sup>]
  - (26) Threshold stress,  $\sigma_{th}$  [psi]
  - (27–32)            unused
- PROPS(33–40)       Inelastic properties of Matrix (shear stresses)
- (33) 1st slope of inelastic strain,  $a_1$  [1/psi]
  - (34) 1st threshold stress,  $\tau_{th1}$  [psi]
  - (35) 1st stress width,  $d\tau_{th1}$  [psi]
  - (36) 2nd slope of inelastic strain,  $a_2$  [1/psi]
  - (37) 2nd threshold stress,  $\tau_{th2}$  [psi]
  - (38) 2nd stress width,  $d\tau_{th2}$  [psi]
  - (39–40)            unused
- PROPS(41–56)       Fatigue and Creep Parameters

FIGURE 29. CMCUMAT DAMAGE MATERIAL PROPERTY INPUTS

## 4.2. HEAT TRANSFER ANALYSIS INPUT

The heat transfer input file is file code 32, known as the variable NT32 in the CSTEM code. This section contains a general description of the heat transfer input and how it is used. Section 5.2. contains an explicit summary of these input variables and the order in which they occur in the input deck.

### 4.2.1. Heat Transfer Analysis Types

There are four different types of heat transfer analysis that can be performed with CSTEM. They are linear steady state, nonlinear steady state, linear transient, and nonlinear transient. Section 3.2. describes these types of heat transfer analysis in detail. A nonlinear heat transfer analysis contains temperature dependencies and a transient heat transfer analysis contains time dependencies. The specification of the type of heat transfer analysis is contained in the analysis control and structural input deck. Most of the inputs contained on the heat transfer input deck vary according to the type of heat transfer analysis being performed.

### 4.2.2. Structural Coupling

A heat transfer analysis is coupled to a structural analysis by using the heat transfer results in terms of nodal temperatures in the structural solution of a problem. It is possible to perform a standalone heat transfer analysis. Several input variables that apply to a heat transfer analysis are contained on the analysis control and structural input deck, file NT2. The geometry, heat transfer analysis type, coupled solution points, and any initial nodal temperatures for the heat transfer analysis are contained on the analysis control and structural solution input deck. Details concerning these variables can be found in Section 4.1.

### 4.2.3. Heat Transfer Load Case Control Information

Section XXVII. of the input sheets contains information that controls the subsequent reading of heat transfer material properties and boundary conditions, as well as the specification of time and temperature steps in a particular heat transfer load case. The point at which a heat transfer



analysis is to be restarted from a previous partial analysis is also indicated by Section XXVII.  
input.

#### 4.2.3.1. Time Step and Temperature Points

There are four types of heat transfer analyses: Linear Steady State, Nonlinear Steady State, Linear Transient, and Nonlinear Transient. Nonlinear analyses must have heat transfer material properties defined at multiple temperatures while transient analyses require specification of boundary conditions at multiple time points. Of the four types of heat transfer analyses only type 1 (Linear Steady State) does not need to specify any time or temperature points.

A transient heat transfer analysis is done by stepping along in time and obtaining thermal equilibrium at each calculation point. A time step is the period of time between two time step points. The variable NTIMSP indicates the number of time step points that are to be read. Generally time steps are entered as hours, but a negative value of NTIMSP signals that the time steps are entered in units of seconds. The actual time step values may be entered on as many lines as necessary, but there is a limit on the number of time steps allowed in a single load case. This limit is controlled by the parameter MXTIM and is currently set at 20.

The time at the beginning of a transient heat transfer load case is indicated by the first time step point. For the first load case this is usually a zero. The transient heat transfer analysis load case will continue until the time entered as the last time step point is reached. Intermediate time step points are used to change any thermal boundary conditions, print out results for the just completed time step, and as points in time where another type of analysis solution may be desired.

Each time step is divided into a number of equal time subincrements. The number of subincrements in each time step are entered immediately following the time step points. More than one line may be used to enter the time subincrements. The analysis proceeds one time subincrement at a time, obtaining thermal equilibrium at each time subincrement point. Thermal boundary conditions at each time subincrement point are obtained by linearly interpolating between the value of the boundary condition at the beginning and ending points of the time step.

Nonlinear heat transfer analyses are performed using an iterative procedure in which the heat transfer material properties are formed using the temperature results of the previous iteration and continuing until convergence is achieved to some tolerance. The material properties are entered at various temperatures and are linearly interpolated between the input temperature values. The temperatures at which the material properties are entered may vary from material to material, but should span the expected resultant temperatures from the heat transfer analysis. There is a limit on the number of temperatures at which the thermal material properties can be defined. This limit is controlled by the parameter MXHTMP and is currently set to 20.

#### 4.2.3.2. Analysis Options

Section XXVII.1. of the input sheets describes the analysis options for a heat transfer analysis. This is used to activate various boundary condition types so that they are read from the input deck and included in the analysis. Convergence criteria, restart indication, and integration order of the heat transfer finite elements are also indicated on this line.

Heat transfer boundary conditions will be included as indicated by the analysis options. Basically, a zero indicates the particular type of boundary condition is not to be included and a one indicates that it is to be included in the analysis. If a boundary condition type is to be included, some type of information on that boundary condition will have to be input as described in later sections. Since the boundary conditions and material property arrays are not reinitialized at each load case a shortened format can generally be used to indicate if the boundary condition information is unchanged from the previous load case.

For nonlinear heat transfer analyses, an iteration procedure is used to arrive at a solution. The convergence criteria for a heat transfer analysis is based on the ratio of second vector norms, where the second vector norm is defined as the square root of the sum of each vector term squared. The second vector norm of the current iteration temperature increment divided by the second vector norm of the resulting temperature for the current iteration must be less than the input convergence value. If the convergence value indicated on the input deck is less than 1.E-8, then the default value of 1.E-7 will be used. The maximum number of iterations allowed before execution is

stopped must be indicated; there is no default for this input. If convergence is not achieved before the indicated number of iterations, the current heat transfer solution will be printed and execution stopped.

#### 4.2.3.3. Restart

Analysis restart from a previously converged timestep is possible for transient nonlinear heat transfer. A restart file containing the nodal temperature solution for the latest converged timestep is written for the transient nonlinear analysis case. Timestep points only are written and only the latest timestep point is retained; the restart file is rewound before writing a converged solution so timestep solutions previous to the last converged timestep are not saved. The restart is done by reading the input deck to get the proper thermal boundary conditions and time steps, and reading the heat transfer restart file to get the current temperature distribution. The restart case from which the solution is to proceed is found by matching the time step number and time of the converged timestep stored on the restart file with the values indicated on the input deck.

#### 4.2.4. Material Properties

Heat transfer material properties include mass density, thermal conductivity, specific heat, and radiation emissivity. Thermal conductivities for each material included in the heat transfer model must be input to perform the analysis. Mass density and specific heat describe what may be viewed as the thermal inertia of the model and so are required for transient analyses. If not specified, the mass density will be calculated from the structural density specified for the same material in the analysis control input deck. Radiation emissivity is required for radiation heat transfer analyses.

The number of materials in the model is indicated in section I.2. of the input sheets as part of the analysis control deck. Material properties are entered for a set of temperatures for each material. It is not necessary that the same temperatures be used for each material. The material number to which the properties apply is indicated before the properties themselves. Normally, material properties are read until input for the total number of materials has been found. A zero

entered as the material number indicates the end of material property input if not all materials are needed to be input, such as in subsequent load cases.

#### 4.2.5. Data Sets

Many of the boundary condition specifications for heat transfer analysis make use of data sets. A data set for a particular quantity contains values of that quantity at all of the time steps or temperature points in the heat transfer load case. The various quantities which use data sets are internal heat generation, convection coefficients, convection temperatures, surface heat flux, nodal heat flux, prescribed nodal temperatures, radiation temperatures, and radiation emissivity. Of these quantities, only radiation emissivity is specified at various temperature points. All the other quantities are specified as a function of the time steps.

Data sets for a particular quantity are numbered and assigned to elements or nodes by reference to this number. Currently, up to 100 data sets can be specified for a particular quantity as controlled by the parameter MXSET. From load case to load case, a data set can be completely respecified, scaled, or incremented. The scaling and incrementing feature is particularly useful if a distribution is to be maintained, but the actual values are to be changed.

#### 4.2.6. Internal Heat Generation

Internal heat generation is an elemental quantity. It is a heat generation rate per unit volume that is added to an element thermal load vector and then assembled into the total thermal load vector of the model. The amount of heat generated is constant within an element. Section XXX. of the input sheets pertains to specification of internal heat generation. This section is entered only if the analysis option variable NHGEN was entered as 1. The first section of XXX. has to do with the assignment of a data set number to various elements. Note that several elements can be tagged with a given data set number at one time by making use of the beginning and ending element number and element number increment. These can be thought of as counters in a DO loop to tag elements with the data set number. A single element would be specified by indicating the element to be both the beginning and ending element number. The end of this input section is signalled by a zero element number. If internal heat generation is to be included in the load case, but the element

specification is the same as in the previous load case a zero in this section is all that is needed to signal this.

The second section of XXX. loads the data sets themselves with the heat generation rate per unit volume at each time step point. A negative value of heat generation would indicate a heat sink. The data set can be overwritten by these values, or the values can be used as scale factors or increments to be applied to the data set. Multiple data sets can be specified at one time using the beginning and ending data set number and data set number increment, similar to the element specification. The end of the input for this section is indicated by a zero data set number. If internal heat generation is to be included in the load case and the data set values are the same as in the previous load case a zero is all that would be needed to keep the data sets the same.

#### 4.2.7. Convection Heat Transfer

Convection heat transfer is specified by element and face number. The convection boundary conditions contribute to the thermal load vector on the model, similar to a structural pressure load. Convection also contributes to the conductivity matrix as shown in equation (122) in order to account for the dependence of convection heat flux on the temperature difference between the environment and the model. The face number is determined by the element connectivity in the same way as for pressure boundary conditions. An element may have more than one face with convection boundary conditions. Section XXXI. of the input sheets pertains to specification of convection boundary conditions. This section is entered only if the analysis option variable NCBC was entered as 1. The element and faces are specified by assigning a convection boundary condition data set to the element face. Several elements having the same convection boundary condition on the same face number can be specified at the same time by using the beginning, ending and increment in element number. A zero for element number indicates the end of convection element specification input.

Convection environment temperature and convection coefficient values for a data set specification are entered on separate lines, with the environment temperature specified first then the convection coefficient. The convection temperature and convection coefficient data sets are input at each time step point. The actual values can be input to overwrite any previous values for the

data set, or scale factors or increments can also be input. A zero data set number indicates the end of input for each of these data sets.

#### 4.2.8. Surface Heat Flux

Section XXXII. of the input sheets pertains to surface heat flux input. This section is entered only if the analysis option variable NFBC was entered as 1. Surface heat flux is integrated over the applied element face and is assembled into the thermal load vector, similar to a pressure force for a structural model. The input for surface heat flux is very similar to that for convection heat transfer. Surface heat fluxes are applied by tagging element and face numbers with a data set number. Several element faces can be specified at once by using the beginning, ending, and increment in element numbers. A zero for element number indicates the end of surface heat flux element specification.

The surface heat flux data set values are specified at each time step point. Similar to other types of data set entries these values can be used to overwrite any previous values, as scale factors for previous values, or as increments to be applied to previous surface heat flux data set values. Several data sets can be modified at a time using the beginning, ending, and increment feature. A zero data set number indicates the end of this input.

#### 4.2.9. Nodal Heat Flux

Section XXXIII. of the input sheets pertains to specification of nodal heat flux. This section is entered only if the analysis option variable NFLX is set to 1. Nodal heat flux represents a heat rate entering or exiting the model through a node. This is similar to a point load in a structural model. Nodal heat fluxes are applied to nodes by tagging the node with an appropriate nodal heat flux data set number. Several nodes with the same nodal heat flux can be specified at one time by making use of the beginning, ending, and increment in node number. A zero node number is used to indicate the end of nodal heat flux nodal specifications.

The actual nodal heat flux values are assigned data set numbers in the next section. Values are entered for each time step point. These values can be actual heat flux values to overwrite any existing heat flux values stored in the data set from a previous load case, or they may be scale

factors or increments to be applied to any previous values. A zero data set number indicates the end of this input.

#### 4.2.10. Prescribed Nodal Temperatures

Section XXXIV. of the input sheets pertains to input of prescribed nodal temperatures. This section is entered only if the analysis option variable NPRES is entered as 1. Prescribed nodal temperatures are similar to prescribed nodal displacements in a structural analysis, and may be necessary to avoid an unrestrained conductivity matrix. Prescribed nodal temperatures may not be needed when using only true heat transfer boundary conditions such as convection and radiation. They will most probably be necessary when using a derived heat transfer boundary condition such as heat flux and heat generation. Prescribed nodal temperatures make use of data sets like many other heat transfer boundary condition inputs. The nodes with prescribed nodal temperatures are assigned a data set number with the possibility of specifying several nodes using the beginning, ending, and increment in node number. A zero node number indicates the end of the node specification of prescribed nodal temperatures.

The actual prescribed nodal temperature is specified at each time step point and assigned to a data set. These may be actual values which overwrite any previous nodal temperature values in the data set, or they may be scale factors or increments which are applied to the previous value stored in the data set. A zero data set number indicates the end of this input.

#### 4.2.11. Radiation Heat Transfer

Input for radiation heat transfer boundary conditions are described in section XXXV. of the input sheets. Radiation heat transfer is available for nonlinear analyses only. This section is entered only if the analysis option variable NRBC is set to 1. Radiation boundary conditions are similar to that for convection boundary conditions both in function and input. Functionally, radiation boundary conditions contribute to both the thermal load vector and to the conductivity matrix of the model as shown in equation (129). This is due to the dependance of radiation heat transfer on the temperature difference between the environment and the model. The radiation heat transfer input requires two quantities be specified. These quantities are environment temperatures

and emissivity scale factor which are assigned together for a particular data set specification, but entered on separate lines, similar to convection input. Gaseous emissivities and absorptivities will be required if gaseous radiation is to be calculated. If the gaseous emissivity and absorptivity for a particular radiation data set are absent (or zero), surface to surface radiation is assumed for that data set.

The radiation boundary conditions are assigned to an element and face by tagging the face with a radiation data set number. A face of several elements can be tagged at one time by specifying a beginning, ending, and increment in element number. A single element is specified by either entering the element number as both the beginning and ending number, or by omitting the ending element number. A zero entered for the element number indicates the end of the element specification for radiation boundary conditions.

The radiation temperatures are entered at each time step point. On a separate line following the radiation temperatures the emissivity scale factor and, optionally, the gaseous emissivity and gaseous absorptivity is entered on a single line. The values entered for any of these quantities can be used to overwrite the previous values, or as scale factors or increments to be applied to the previous values. A zero data set number ends the radiation data set input.

The surface emissivity values are entered as material properties for a sequence of temperatures. An emissivity scale factor can be used to multiply these emissivity values, providing a means to spatially vary the emissivity. The emissivity scale factor along with the gaseous emissivity and gaseous absorptivity can also be a function of temperature. The temperatures at which these values are defined are those for the material of the element or of the surface layer if the element is layered. A separate line is entered for each temperature to define the emissivity scale factor and gaseous emissivity and gaseous absorptivity.

Gaseous emissivity and gaseous absorptivity values are required if gaseous radiation is to be performed. These quantities are entered for the data sets which correspond to gaseous radiation on the same line as the emissivity scale factor. If left unspecified or set to zero, these quantities take on the default value of 1.0, which corresponds to surface to surface radiation. Gaseous emissivity and gaseous absorptivity can be functions of temperature as described in the previous paragraph.



### 4.3. ELECTROMAGNETIC ANALYSIS INPUT

The input deck for electromagnetic analysis is contained on file code 52. In the CSTEM code, the variable NT52 contains this file code. This section gives a general description of the electromagnetic input and its use. A detailed description of the electromagnetic input variables and their exact input order can be found in section 5.3.

Three different methods can be used in the analysis of electromagnetic absorption in CSTEM. These methods are the wave matrix method, the optics method, or a table lookup method. The method to be used is defined in the structural input deck as the variable IEMAG, described in section I.4. of the input sheets.

#### 4.3.1. Material Description

The electromagnetic material properties are entered through the use of a data bank. Each set of data in the data bank has an alphanumeric material identifier to which it is associated. The material description input matches the material identifier with the various material numbers that have been assigned to the geometry of the finite element model. Each material number must be matched with a material identifier, although the same material identifier may be used for more than one material number. By entering the material number as negative, the data bank properties associated with the material identifier will be printed on the input echo file, NT1.

A special material identifier, COND, is available for materials which are conductors and so reflect all incident radiation. If a material with this identifier is encountered in the path of a propagating electromagnetic wave, the wave is assumed to be completely reflected. The COND material identifier would be used for all types of metallic materials.

The matching of material identifiers with the material numbers used in the analysis needs to be done only once, so this input is entered only once in the electromagnetic input deck following the title line.

#### 4.3.2. Electromagnetic Wave, Path, and Load Case Specifications

The electromagnetic analyzer of CSTEM examines a specific electromagnetic wave propagating along a specific path through the model. Several waves and several paths may be defined with an electromagnetic load case defined as one combination of a specific wave and path.

The electromagnetic waves are defined by their orientation, magnitude, polarization angle, and frequency. These quantities are read on one line of input so that each line describes a specific wave. The number of input lines to be read is indicated by the variable, NWAVE.

The orientation of the wave is specified by use of a transformation number. The actual transformation associated with the number is defined in the structural input deck using one of the methods as described in section VIII. of the input sheets. The coordinate system used to define the electromagnetic wave is shown in Figure 5. The magnitude of the wave is used in the optics and table lookup methods only, and is usually entered as a relative value of 1 or 100 to indicate that the wave is at an impinging strength. The polarization of the electromagnetic wave is used only in the table lookup method, in which the wave polarization angle is used to linearly interpolate between the polarization angles in the absorption data bank. The wave frequency, entered in hertz, is probably the most important of the properties characterizing the electromagnetic wave. The frequency and orientation of the wave are the two quantities which are necessary for the primary electromagnetic analysis method, the wave matrix method. A maximum number of waves may be specified depending on the value of the parameter MAXW. Currently MAXW is set to 10.

The propagation path of the impinging electromagnetic wave is initialized by defining the impingement point of the wave. This consists of specifying the element number and surface face number at the impingement location. The stacking sequence of materials encountered by the propagating wave is then determined by progressing through the cross section of the model from the surface until the opposite surface is reached or a conductor material is encountered which will totally reflect the wave. A maximum number of paths may be specified depending on the parameter MAXP, which is currently set at 10.

An alternative to specifically specifying the impingement location is to have the code perform an exposure analysis of the model from the perspective of the incoming wave. By using a

hidden line technique as done in computer graphics, the faces of the model which can be seen by the incoming wave are determined. The exposure of a face is determined by the amount of the face edges that are visible to the incoming wave. All faces exposed more than 90% will then be used as an impingement point for a propagation path.

Finally, the number of load cases in the absorption analysis are entered followed by the wave and path combination to be used in each load case. The waves and paths are numbered consecutively in order of input. If an exposure analysis is indicated in the path description, this will automatically be performed for every defined wave. If only the exposure analysis is to be used to define the load cases, the number of load cases would be entered as zero.

#### 4.3.3. Data Bank Formats

There are three different formats in which the electromagnetic material properties can be entered in the data bank. Two of these formats can be used with either the wave matrix method or the optics method of electromagnetic analysis, while the third format is used with the table lookup method only. These different formats may not all be contained in the same data bank. The two formats for the wave matrix and optics method can be contained on the same data bank, but the table lookup data bank format must exist on a separate data bank. If nonunique material identifiers are used, the first encountered occurrence will be used to define the material. Note that the temperatures, frequencies, and angles should be entered in increasing order.

In all three formats, the entry for a material begins with the material identifier, an alphanumeric string of from one to eight characters with no spaces. On the same line as the identifier is an integer which determines the number of different temperatures for which properties are contained on the data bank. The next line contains the first of these temperatures and an integer defining the number of frequencies at that temperature for which properties are entered.

The line following the temperature is where the different data bank formats begin to vary. For the wave matrix and optics formats, the next lines contain the frequency (in megahertz) at the current temperature and either the real and imaginary parts of the permittivity and permeability, or optionally the dielectric constant and loss tangent, of the material at that temperature and frequency. After the number of frequencies equal to the integer value on the temperature line have

been read, another temperature line is expected. This continues until the total number of temperatures equal to the integer value on the material identifier line have been read.

In the table lookup data bank format, the line after the temperature and number of frequencies to be read contains a frequency (in megahertz) and an integer describing the number of polarization angles at that frequency for which data is contained. The next line contains a polarization angle and an integer which is generally zero or one. This integer can be used for expansion of the data bank to contain an additional interpolation parameter, but is not currently used now. After the polarization angle line is a line which contains two values. The first is not used and is the slot available for data bank expansion. The second value is a decimal amount of energy that would be absorbed by the material at the current temperature for a wave at the current frequency and polarization angle. The next line contains the next polarization angle and is followed by an absorption line. This continues until the number of polarization angles expected has been encountered. Another frequency line is then expected, followed by the polarization angles and absorption amounts at that frequency. After all the frequencies have been read, another temperature line is read. This continues until the number of temperatures as indicated on the material identifier line have been read.

#### 4.4. ACOUSTICS ANALYSIS INPUT

The input deck for an acoustics is contained on file code 54. This section gives a general description of the acoustics input and how it is used. A detailed description of the acoustics input variables and the order in which they are input is contained in section 5.4.

An acoustics analysis is activated by the control variable, NOISE, which is contained on the structural analysis and control deck and is described in section I.4. of the input sheets. An acoustics analysis is done in conjunction with a free vibration analysis of the structure, which serves as a secondary actuation for an acoustics analysis on the load case level. This is controlled by the variable IEIG, described in section XI.1. of the input sheets.

##### 4.4.1. Nodal Forcing Function

The acoustics analyzer contained in CSTEM uses a nodal point load as a forcing function to preferentially excite the structural frequencies. A single load is applied at a node specified by the variable NODFRC in the global direction indicated by the variable NODDIR. The acoustics analysis is performed at each calculated structural frequency for a range of forcing frequencies which are logarithmically distributed between the input frequencies indicated by FRQMIN and FRQMAX. The amplitude of this forcing function is input using the variable FORCE. If a zero value of FORCE is indicated, an unforced solution for sound power is calculated.

##### 4.4.2. Modal Loss Factor

The modal loss factors are input one for each structural frequency calculated and indicate the vibrational damping of the structure. These modal loss factors can only be easily calculated for systems containing some type of additive damping systems. For a structure without additive damping the modal loss factors can only be estimated based on previous data derived from tests or from a similar structure.

##### 4.4.3. Masking Point

Some structures may be of such a configuration that certain free faces may not be desired to be included in the acoustic analysis. Such a situation may occur in the analysis of a duct in which

the faces on the inner surface of the duct are not desired as part of the acoustic analysis. In such an instance a masking point can be specified so that any free surfaces that can see the masking point will not be included in the acoustic analysis. Several masking points up to a limit set by the parameter MDRMSK can be used. The current limit is 5. The masking points are defined by specification of the global coordinates of the point.

#### 4.4.4. Results of Acoustic Analysis

The results of an acoustic analysis are given in terms of radiation efficiencies and sound power. These quantities are specified at each structural frequency for a range of forcing frequencies. Additionally, the sound power is summed over all the structural vibration frequencies. The radiation efficiencies allow a relative measure of the structural modes and forcing frequencies which contribute to sound radiation. The sound power is a direct measure of the generated acoustic pressure. The sound power is given in units of watts (1 watt = 8.8507 (in-lb<sub>f</sub>)/sec and can be converted into intensity,  $I$ , by dividing by an area upon which the acoustic pressure is impinging. The root mean square of the acoustic pressure,  $\bar{P}$ , can be found as  $\bar{P} = \sqrt{\rho c I}$  where  $\rho$  is the density of the medium through which the sound is propagating and  $c$  is the speed of sound in that medium.

## 4.5. TAILORING INPUT

The input deck for control of the tailoring process is contained on file code 75. This section contains a general description of this input and some recommendations as to how it is used. A specific description of the input variables and the order in which they are input is contained in section 5.5.

### 4.5.1. Input Form

The tailoring procedure used in CSTEM is a version of COPES/CONMIN that was extracted from the NASA computer code, STAEBL. Very little has been changed from this application in the tailoring portion of the code itself as well as in the controlling input. The input is read by the tailoring module routine, COPE08, as opposed to the general read routine, READZR, that is used throughout the rest of the CSTEM code. The major differences in recognized input options on the tailoring input deck is that the dollar sign (\$) comment designator must occur in column 1 and that input data is examined to determine whether it is in formatted or unformatted form. A line of input is considered unformatted if a comma is found on the line or if all data occurs within the first 10 columns of a line. The tailoring control input contained on file 75 is unformatted, so a comma should be used as a delimiter between the input fields. A space can be used as a delimiter between fields, but if the last non-blank character on a line of input extends beyond column 10 an error in interpreting the input may occur. For this reason it is recommended that commas instead of spaces be used as delimiters.

### 4.5.2. Procedure Type

The very first line on the tailoring input deck is a title line used for identification. Immediately following the title a line is entered which begins with a dollar sign (\$) and is followed by an integer which indicates the type of tailoring procedure to perform. This integer is used within the code to identify the particular subroutine to be called from the routine ANALIZ so that the tailoring variables with the proper elements from the tailoring global common, GLOBCM, so that the desired tailoring analysis will be performed. This process is somewhat problem dependant,

requiring new subroutines to be included in the code if a different type of tailoring process is to be performed.

#### 4.5.3. Control Variables

The two lines after the specification of the tailoring procedure are used for control of the tailoring process. Whether the run is to simply read the input as an input check, cycle once through the tailoring process, or perform a complete tailoring analysis is indicated by the variable NCALC. On the same line as NCALC, the variable NDV indicates the number of design variables that are to be included in the analysis. These are variables which the tailoring program is allowed to change in order to affect the solution and tailor the objective function.

The next control line determines the frequency and amount of printout, the allowable number of iterations, a conjugate direction restart parameter, a scaling parameter, and the number of consecutive iterations in which the convergence criterion are to be satisfied before the solution is considered to be converged. In general, the final three variables on this line (ICNDIR, NSCAL, and ITRM) are given default values. The number of allowed iterations should be chosen with care since the cost per iteration may be expensive if the problem is large with many design variables and constraints.

#### 4.5.4. Tolerances

The next two lines contain a number of tolerances and change limits. These are generally left at default values unless the problem is fairly well characterized and the tailoring procedure is known to progress in the desired direction. If convergence is being delayed by change limits which are too small, an increase in the variables FDCH, DELFUN, and/or ALPHAX may be called for. Conversely, if an oscillation in the solution is perceived it may be desirable to tighten these tolerances.

#### 4.5.5. Object and Design Variables

The next three sets of inputs pertain to the specification of the object function, that value which is to be tailored to obtain a minimum or maximum, and the design variables. The variable NDVTOT is generally entered as the number of design variables, NDV. The variable IOBJ is an



integer which gives the location in the tailoring global common, GLOBCM, in which the calculated object function value is stored. Whether this object function is to be maximized or minimized is indicated by the variable SGNOPT. Table 6. gives a list of variables which are currently stored in GLOBCM. The variables in locations 1 – 71 are stored from the access to the executive through subroutine CSTEM, while the remaining variables are stored from either specialty subroutines called from the stress recovery and output phase of the finite element portion of the code or from the tailoring "ANALIZn" routines.

Upper and lower bounds for each design variable and an initial value for each design variable is entered next. The order of input is in increasing consecutive order beginning with design variable 1. The next set defines the design variable numbers and the location in global common where the values of the particular design variable is stored. The variable AMULT is a multiplier that can be applied to the design variable value.

#### 4.5.6. Constraints

Constraints are limits that are not to be exceeded. The number of these constraints are indicated by the variable NCONS. The next two lines are entered in pairs, one pair for each constraint. These lines define the location in global common of the constrain value, and the lower and upper bound of the constraint. Scale factors to be applied to the constraint bounds can be input if desired. Following the constraints is a line with END to indicate the end of the tailoring input.

#### 4.5.7. Current Tailoring Procedures

Tailoring in CSTEM may require the user to provide an "ANALIZn" routine which is called by the subroutine ANALIZ based on the procedure number specified on the tailoring input deck. The "ANALIZn" routine assigns the design variable changes resulting from the tailoring process to the proper locations in common (or possibly on file) for use in the subsequent finite element evaluation. It may also assign the object and constraint variable values resulting from the finite element evaluation to the proper locations of the tailoring common GLOBCM if these are not one of the preassigned variables. This "ANALIZn" routine is generally quite simple. The template

shown in Figure 30. can be followed in creating this routine. The currently implemented tailoring procedures are listed in Table 7.

A single calling argument, ICALC, is passed into the "ANALIZn" routine. The "ANALIZn" routine should contain the commons GLOBCM, OPLAY, and the common MFBANK if ply constituent properties are to be used as a design variable. In addition to these commons, the various PARAMETER statements associated with these commons should also be included. A local variable IPATH=ICALC-1 should be immediately defined, followed by a branch on the variable ICALC. This branch is based on whether the "ANALIZn" routine is being entered for the first time (ICALC=1) or a subsequent time. The first entry to "ANALIZn" will skip any assignment of design variables.

The layer thicknesses and angles contained in common OPLAY are generally the values used as design variables. By reassigning these values with the tailored design variables, the model can be relayered with the tailored thicknesses and/or angles. To accomplish this relayering the steps are:

- 1) Assign the design variables to the appropriate locations in THK and ANGLAY
- 2) Set local variables ILYRN=0, IRESGN=2, and IGAX= the value used for the particular problem
- 3) CALL LAYUP (ILYRN, IRESGN, IGAX)
- 4) CALL RESTAR
- 5) The next evaluation can then be performed with the relayered model (i.e. CALL CSTEM (IPATH) )

```

SUBROUTINE ANA(ICALC)
C
C _____
  PARAMETER ( NNMAX=0000,NEMAX=000,NO=NNMAX,MNDOF=3*NO,MDOFE=66,
&MDOFN=3,MNPE=20,NTEMP=10,NPSMAX=10,NCONMX=5,MFACE=6,MPNODE=8,
&LAYMX=00)
C
  PARAMETER (MAXGEN=25)
C
C _____
  COMMON /OPLAY/ LMAT(MAXGEN,LAYMX), THK(MAXGEN,LAYMX),
&ANGLAY(MAXGEN,LAYMX), LTRN(MAXGEN,LAYMX), NLAY(MAXGEN), NGEN,
&IGEN(LAYMX,MAXGEN), NCROSS, LCROSS(6,NEMAX)
C
C _____
  COMMON /GLOBCM/ AR(200)
C
C _____
  IPATH=ICALC-1
C
  IF(IPATH.EQ.0) GOTO 10

```

*In this location, place design variables in OPLAY or other finite element variables as needed to perform a finite element evalutaion.*

```

  ILYRN=0
  IRESGN=2
  IGAX=0
C
C Form Element Layering
C
  CALL LAYUP (ILYRN, IRESGN, IGAX)
C
C Reinitialize Int.Pt. Record Numbers
C
  CALL RESTAR
C
C Perform Analysis, Which Will Put Preassigned Values In GLOBCM
C
  10 CALL CSTEM (IPATH)

```

*In this location, place any unassigned object or constraint variables in GLOBCM as needed to perform a tailoring evalutaion.*

```

  RETURN

```

FIGURE 30. TEMPLATE FOR THE "ANALIZn" ROUTINE

TABLE 6. TAILORING GLOBAL COMMON

LOCATION IN /GLOBCM/	VARIABLE
1 – 8	Vibration Frequencies
9 – 16	Ply Orientation Angles
17 – 24	Ply Thickness Fractions
25	TE Electromagnetic Absorption
26 – 33	Sound Power From Each Free Vibration Frequency
34	Cost
35 – 42	Volume Fraction
43	Total Weight
44	Max Nodal Temperature
45	Max Temperature Difference
46 – 69	Layer Generation Set Thicknesses
70	Max Mode Shape Slope
71	TM Electromagnetic Absorption
72 – 75	unused
76	Cross Sectional Thickness
77 – 79	unused
80 – 85	Max Total Strain Components (local or global according to IPOUT)
86 – 91	Min Total Strain Components (local or global according to IPOUT)
92 – 97	Max Stress Components (local or global according to IPOUT)
98 – 103	Min Stress Components (local or global according to IPOUT)
104	Max Interlaminar Stress (if local requested using IPOUT)
105 – 109	unused
110 – 112	Max Global Displacement Components
113 – 115	Min Global Displacement Components
116 – 118	Max Skew Displacement Components
119 – 121	Min Skew Displacement Components
122 – 200	unused

TABLE 7. CURRENT TAILORING PROCEDURES

ROUTINE	GEOMETRY	OBJECT	DESIGN	CONSTRAINT
ANALIZ1	4 layers, constant thickness model	EM absorption	3 ply thicknesses	
ANALIZ2	4 layers, constant thickness model	1st natural freq	4 ply angles	2nd and 3rd freqs
ANALIZ3	4 layers, constant thickness model	sound power	4 ply angles	
ANALIZ4	4 layers, constant thickness model	cost (based on fiber)	3 ply thicknesses, 4 FVR	
ANALIZ5	4 layers, constant thickness model	weight	3 ply thicknesses, 4 FVR	
ANALIZ6	4 layers, constant thickness model	max nodal temperature	3 ply thicknesses, 4 angles, 4 FVR	
ANALIZ7	4 layers, constant thickness model	max temperature difference	3 ply thicknesses, 4 angles, 4 FVR	
ANALIZ8	1 generation set of 8 layers max	1st natural freq	ply thicknesses, ply angles	2nd and 3rd freqs
ANALIZ9	1 generation set of 8 layers max	max modeshape slope	ply thicknesses, ply angles	2nd and 3rd freqs
ANALIZ10	multi-generation sets, 8 layers each	EM absorption	ply thicknesses	
ANALIZ11	1 generation set of 8 layers max	weight	ply thicknesses, ply angles	
ANALIZ12	multi-generation sets, 8 layers each	weight	ply thicknesses	1st natural freq
ANA1 (13)	8 layers, constant thickness model	thickness	3 ply thicknesses	min/max strains
ANA1 (14)	2 angle-ply model	interlaminar stress	ply angle	

#### 4.5.8. Layered Tailoring

Tailoring of layered models can be easily accomplished using the automatic layering features of CSTEM. This is done through the use of the layering variables in common OPLAY. Common OPLAY appears as listed here:

```
COMMON /OPLAY/ LMAT(MAXGEN,LAYMX), THK(MAXGEN,LAYMX),  
&ANGLAY(MAXGEN,LAYMX), LTRN(MAXGEN,LAYMX), NLAY(MAXGEN), NGEN,  
&IGEN(LAYMX,MAXGEN), NCROSS, LCROSS(6,NEMAX)
```

The array dimensions of interest in OPLAY are MAXGEN and LAYMX. The MAXGEN dimension refers to the generation set number and the LAYMX dimension refers to the number of layers in the particular generation set.

The variables in OPLAY are as follows:

LMAT(I,J)	Material number for layer J within generation set I
THK(I,J)	Thickness in inches for layer J within generation set I
ANGLAY(I,J)	Angle in degrees for layer J within generation set I
LTRN(I,J)	Transformation number used in lieu of angle in ANGLAY
NLAY(I)	Number of layers in generation set I
NGEN	Number of generation sets
IGEN(J,I)	Order J of generation sets I
NCROSS	Number of cross sections to be assigned
LCROSS(K,L)	Cross section assignment parameters for cross section L
where	
K=1	Beginning element number of first cross section
K=2	Through thickness axis
K=3	Generation order
K=4	Beginning element number of ending cross section
K=5	Beginning element number increment
K=6	Optional cross section ending element number

## **SECTION 5. INPUT SHEETS**

The following pages are summary descriptions of the input variables. This is probably the section to be directly referenced when creating an input deck to run in CSTEM. Inputs for the various analysis modules are generally contained on different files to ease the activation and deactivation of the various analysis types in a problem. This section is divided in the same way as the different analysis type input decks. Refer to section 4. for a more detailed description of the input and its meaning. A comprehensive outline of the CSTEM input sheets follow.

## **CSTEM INPUT SHEETS**

### **ANALYSIS CONTROL AND STRUCTURAL INPUT**

#### **I. HEADING AND CONTROL INFORMATION**

- I.1. Title Card
- I.2. Problem Size Data
- I.3. Restart and Output File Creation Options
- I.4. Analysis Options

#### **II. NODAL COORDINATES**

#### **III. NODAL SKEW COORDINATE SYSTEMS**

- III.1. Input Skews
- III.2. Pregenerated Skews

#### **IV. CONSTRAINT EQUATIONS**

#### **V. ELEMENT DEFINITIONS**

- V.1. Header Line For Each Element Type
- V.2. 8 Noded Solid
- V.3. 16 Noded Solid
- V.4. 20 Noded Solid
- V.5. 4 Noded Shell
- V.6. Mesh Generation
  - V.6.a. Thickness Specification
  - V.6.b. Rotation #1 Specification
  - V.6.c. Rotation #2 Specification
  - V.6.d. Layer Orientation Definition
  - V.6.e. Layer Material Definition
  - V.6.f. Generation Order

#### **VI. ELEMENT SKEW MATERIAL COORDINATE SYSTEMS**

#### **VII. ELEMENT LAYER DEFINITIONS**

- VII.1. Manual Layer Input
- VII.2. Cross Section Layup Generation



- VII.2.a. Layer Specification
- VII.2.b. Layup Generation
- VII.2.c. Cross Section Specification

## VIII. SKEW COORDINATE SYSTEM DEFINITIONS

- VIII.1. Reference Nodes
- VIII.2. Direction Cosines
- VIII.3. Euler Angles

## IX. FIXED BOUNDARY CONDITIONS

## X. STIFFNESS ADDITIONS

The remaining groups are used to define or modify materials, loads, displacement boundary conditions, etc. for each load case.

## XI. LOAD CASE CONTROL INFORMATION

- XI.1. Load Case Control Cards

## XII. KEYWORD INPUT

- XII.1. Eigenanalysis Solution Parameter Keyword: EIGS
- XII.2. Alternate Stress Stiffening Keyword: RDKS
- XII.3. Large Displacement Buckling Keyword: LDBK
- XII.4. Plasticity Debug Print Keyword: PDBG
- XII.5. Damage Debug Print Keyword: DDBG
- XII.6. Damage Acceleration Keyword: DACC
- XII.8. Creep Debug Print Keyword: CDBG
- XII.9. Vibration BC Change Keyword: VIBC
- XII.10. Newton–Raphson Keyword: NRIT
- XII.11. Nonlinear Force Calculation Keyword: METH
- XII.12. Microstress Plot Keyword: MICR
- XII.13. Material Selected Stress Summary Keyword: MATX

## XIII. ACCELERATION SPECIFICATION FOR INERTIA OR GRAVITY LOADS

## XIV. MATERIAL ELASTIC PROPERTIES

- XIV.1. Isotropic Material
- XIV.2. Orthotropic Material
- XIV.3. ICAN Generated Elastic Properties
- XIV.4. Failure Criteria Strengths and Parameters
- XIV.5. User Material Properties and Constitutive Data

**XV. PLASTICITY DATA**

**XV.1. Plasticity and Iteration Limit**

**XV.2. Plasticity Options and Convergence Tolerance**

**XV.3. Classical Plasticity Models**

**XV.4. C.T. Sun Orthotropic Plasticity Model**

**XV.4.a. Potential Function Parameters**

**XV.4.b. Master effective Stress–Effective Plastic Strain Data**

**XV.4.b.1. Piecewise Linear Representation**

**XV.4.b.2. Power Law Representation**

**XVI. RATE DEPENDENT INELASTICITY TIME INCREMENT CONTROLS**

**XVII. MATERIAL TIME DEPENDENT PROPERTIES**

**XVII.1. Creep and Iteration Limit**

**XVII.2. Creep Options and Convergence Tolerance**

**XVII.3. Classical Creep Formulation**

**XVII.3.a. Creep Constants**

**XVII.3.b. Creep Rupture Data**

**XVII.4. C.T. Sun Orthotropic Creep Constants**

**XVIII. DAMAGE DATA**

**XVIII.1. Damage and Iteration Limit**

**XVIII.2. Damage Options and Convergence Tolerance Parameters**

**XVIII.3. CMCUMAT Material Data**

**XVIII.4. VPI Polynomial Damage**

**XIX. NODAL DISPLACEMENTS**

**XX. NODAL APPLIED FORCES**

**XXI. NODAL TEMPERATURES**

**XXII. ELEMENT PRESSURE LOADS**

**XXIII. ELEMENT ADDITION/REMOVAL OR MATERIAL CHANGE CODE**

**XXIV. FAILED STIFFNESSES**

**XXV. ICAN ANALYSIS INFORMATION**

## **HEAT TRANSFER ANALYSIS INPUT**

### **XXVI. HEAT TRANSFER TITLE CARD**

### **XXVII. LOAD CASE CONTROL INFORMATION**

XXVII.1. Analysis Options

XXVII.2. Restart Option

XXVII.3. Time Step Points

### **XXVIII. HEAT TRANSFER MATERIAL PROPERTIES**

### **XXIX. HEAT TRANSFER DAMAGE**

XXIX.1. Ply Level Conductivity Damage

XXIX.2. Conductivity Damage Table Temperatures

XXIX.3. Conductivity Damage Tables

XXIX.4. Exponential Damage Law Formulation

XXIX.5. Power Law Damage Formulation

### **XXX. INTERNAL HEAT GENERATION**

XXX.1. Element Specification

XXX.2. Internal Heat Generation Data Sets

### **XXXI. CONVECTION HEAT TRANSFER**

XXXI.1. Element Specification

XXXI.2. Convection Data Sets

### **XXXII. SURFACE HEAT FLUX**

XXXII.1. Element Specification

XXXII.2. Surface Heat Flux Data Sets

### **XXXIII. NODAL HEAT FLUX**

XXXIII.1. Node Specifications

XXXIII.2. Nodal Heat Flux Data Sets

### **XXXIV. PRESCRIBED NODAL TEMPERATURES**

XXXIV.1. Node Specifications

XXXIV.2. Prescribed Temperature Data Sets

### **XXXV. RADIATION HEAT TRANSFER**

XXXV.1. Element Specifications

XXXV.2. Radiation Data Sets

## **ELECTROMAGNETIC ANALYSIS INPUT AND DATA BANK**

**XXXVI. ELECTROMAGNETIC TITLE CARD**

**XXXVII. MATERIAL DESCRIPTION**

**XXXVIII. ELECTROMAGNETIC WAVE DESCRIPTION**

**XXXIX. PROPAGATION PATH DESCRIPTION**

**XL. ABSORPTIVITY LOAD CASE DESCRIPTION**

**XLI. ABSORPTIVITY DATA BANK FORMAT**

**XLI.1. Tables**

**XLI.2. Impedance**

**XLI.3. Dielectric Constant**

## **ACOUSTICS ANALYSIS INPUT**

**XLII. SOUND POWER CALCULATION CONTROL VARIABLES**

**XLIII. FREQUENCY RANGE**

**XLIV. APPLIED FORCE**

**XLV. LOSS FACTORS**

**XLVI. MASKING POINT**

## **TAILORING CONTROL INPUT**

**XLVII. TAILORING TITLE CARD**

**XLVIII. PROCEDURE DEFINITION**

**XLIX. TOLERANCES**

**L. OBJECT AND DESIGN VARIABLES**

**LI. CONSTRAINTS**

**LII. END**

## **5.1. ANALYSIS CONTROL AND STRUCTURAL INPUT**

## I. HEADING AND CONTROL INFORMATION

### I.1. Title Card

#### Line IDENTIFICATION

Up to 80 characters for identification

### I.2.I.2. Problem Size Data

#### Line NN NLTY MMAT NLC INGEOM NBAND IBLK KADD

NN        Number of nodes in the model. (<0 indicates both input and generated nodes)  
NLTY     Number of element types in the model  
MMAT     Number of different materials in the model  
NLC       Number of incremental load cases  
          = 0 Stops after forming elemental stiffnesses  
          < 0 Performs heat transfer and/or absorption analyses only with no structural analysis.  
          Can also be used as a structural input data check, or for cross section interpolation using previously saved integration point constitutive file (FILE8).  
INGEOM   Geometry generation flag  
          = 0 Geometry read in on input file  
          = 1 Geometry pregenerated  
              (contained on 3 files: node-element, transformation, material layer)  
          = -n Generate n surface geometries, coincident nodes will be connected

**NOTE:** If the absolute value of NN does not agree with the number of generated nodes, execution is stopped after generated geometry is printed. Useful for geometry verification.

NBAND    Nodal banding flag  
          =0 No banding (input order is used)  
          =1 Band using GPS banding routine  
IBLK      Solution block sizing flag (may be needed when doing double matrix eigenanalyses like buckling or consistent mass)  
          =0 single block sol'n check made using all of blank common  
          =1 single block sol'n check using half of blank common  
          =-1 all displacements prescribed (stress recovery only)  
KADD      Additional stiffness flag  
          =0 no additions to stiffness matrix  
          =N there will be N stiffness adder matrices input

### I.3. Restart and Output File Creation Options

Line **NOUT NRESTA INREST IPAT ICHK**

**NOUT** Restart output file creation option (used for later restart)

= 0 Do not create restart file

= 1 Create ASCII restart file

= 10 Create binary restart file

= 2 Create ASCII restart file, echo case NRESTA results if restarting

= 20 Create binary restart file, echo case NRESTA results if restarting

= 3 Create binary restart file, read ASCII restart file (no echo)

= 30 Create ASCII restart file, read binary restart file (echo case NRESTA)

NOUT > 0, only last converged load case is saved to restart file

NOUT < 0, all converged load cases saved to restart file

**NRESTA** Restart option

= 0 This is not a restart run

> 0 Load case from which restart proceeds, output from this load case must have been previously written to an output file. Data from load case NRESTA on the output file will be read and becomes the current state from which analysis proceeds. First new load case in current run is NRESTA+1.

**INREST** Next load case in current input deck to be run

**IPAT** PATRAN results file flag

=0 Do not create PATRAN results files

=1 Create ASCII PATRAN results files

=2 Create binary PATRAN results files

=-1 Create PATRAN3 type results files (PATELSIG, PATELEPS, PATDAMGP)

**ICHK** Check run flag

=0 Normal execution

=1 Check run only; stops after layer orientation calculations in 1st load case



#### I.4. Analysis Options

Line **IPLO ICRO IDIS ITCVG CVGTOL IHTR IDMO IEMAG LDORDR NOISE**

- IPLO**      Plasticity option  
            0 = No plasticity analysis  
            1 = Do plasticity analysis using initial stiffness/pseudo force (right hand side) method  
            -1 = Do plasticity analysis using tangent stiffness method
- ICRO**      Creep option  
            0 = No creep analysis  
            1 = Do creep analysis
- IDIS**      Deformation option  
            0 = Small displacement analysis  
            1 = Updated Lagrange large displacement analysis  
            2 = Total Lagrange large displacement analysis (not currently implemented)

**NOTE:** Negative IDIS indicates stress/strain output is with respect to updated coordinates

- ITCVG**    Number of equilibrium iterations allowed. (used for large displacement analysis)  
            >0 stops execution if no convergence  
            <0 continues execution if no convergence
- CVGTOL**   Convergence tolerance on displacement for large displacement equilibrium iterations. Default = .001 used if < 1.E-6 is input
- IHTR**      Heat transfer option  
            0 = No heat transfer analysis  
            1 = Do heat transfer analysis  
            (English units used if IHTR>0, metric units used if IHTR<0)
- IDMO**      Damage analysis flag  
            0 = No damage analysis  
            1 = Do damage analysis using initial stiffness / pseudo force (right hand side) method  
                (Nonlinear Force Keyword of Section XII.11. default = 2 unless explicitly input)  
            -1 = Do damage analysis using tangent stiffness method
- IEMAG**    Electromagnetic option  
            0 = None  
            1 = Do electromagnetic analysis using WAVES  
            2 = Do E/M analysis as single ray light analysis  
            3 = Do E/M analysis using absorption tables
- LDORDR**   Large displacement order  
            1 = (default) 1st order transformation of stress/strain  
            2 = 2nd order transformation of stress/strain
- NOISE**    Acoustic analysis flag  
            0 = No acoustic analysis  
            1 = Do acoustic analysis for load cases that eigenanalysis is requested (see X.1)

*Enter NN lines of II only if INGEOM equals zero*

**II. NODAL COORDINATES**

Line N X Y Z

N	Node number
X	X coordinate of the node [IN]
Y	Y coordinate of the node [IN]
Z	Z coordinate of the node [IN]

### III. NODAL SKEW COORDINATE SYSTEMS

*Enter III.1. only if INGEOM equals zero*

#### III.1. Input Skews

Line **NSKEW**

**NSKEW** Number of nodal skew coordinate systems defined.

*Enter NSKEW of the following lines.*

Line **N NTRN NEND NINC**

<b>N</b>	Node number having a local coordinate system
<b>NTRN</b>	Transformation number defining this skew
<b>NEND</b>	Last node having this local coordinate system
<b>NINC</b>	Increment in node numbers from N to NEND having this local coordinate system

*Enter III.2. only if INGEOM is not equal to zero*

#### III.2. Pregenerated Skews

Line **NSKEWP**

**NSKEWP** Number of pregenerated skew coordinate systems.

*Enter NSKEWP of the following line.*

Line **NBEG NOSKW NEND NINC**

<b>NBEG</b>	Beginning node with this local coordinate system ≠ 0 Beginning node to be skewed = 0 No nodes will be skewed, but transformations will be stored, numbered consecutively.
<b>NOSKW</b>	Skewed node on transformation file which has the transformation matrix.
<b>NEND</b>	Ending node with this local coordinate system
<b>NINC</b>	Increment in node numbers from NBEG to NEND having this local coordinate system

*Enter IV. only if INGEOM equals zero*

#### IV. CONSTRAINT EQUATIONS

Line NCST

NCST      Number of constraint equations to be read

Constraint equation is of the form:  $u_d = C + \sum_i c_i u_i$

*Enter NCST of the following two line sets*

Line NODEP NDPDOF NIND CONST

NODEP      Dependant node number

NDPDOF      Dependant node degree of freedom (1=X, 2=Y, 3=Z)

NIND      Number of independant nodal degrees of freedom to which the dependant nodal degree of freedom is constrained ( i in the constraint equation )

CONST      Constant term in constraint equation ( C in the constraint equation )

*Enter NIND of the following line*

Line COEF INDN INDOF

COEF      Coefficient for the independant nodal degree of freedom ( c in the constraint equation )

INDN      Independant node number

INDOF      Independant node degree of freedom (1=X, 2=Y, 3=Z)

## V. ELEMENT DEFINITION

*Enter NELTYP of the following*

### V.1. Header Line For Each Element Type

Line **NTYPEL NELEMS INCOMP INTORD ISTRP KSAVE LYPRNT**

**NTYPEL** Element type

**NELEMS** Number of elements in this group (not necessary with generated geometry)

**INCOMP** Incompatible modes indicator

0 = Include incompatible modes ( if applicable )

1 = Do not include incompatible modes

-1 = Include incompatible modes without centroid integration modification

**INTORD** integration order ( if applicable )

**ISTRP** print flag, (negative values apply to elem stiffness calculation prints; positive values apply to interpolation of stress/strain from int. pts in unlayered cases)

=0 no elemental printout, int. pt. stress/strain print

=1 print stress/strain at element centroid only

=10 print stress/strain at element nodes and centroid

=100 print stress/strain at element face centers and centroid(also used for

=-1 print element connectivity, integration flags, material number, layering

=-2 print linear elemental stiffness and eqn #'s

=-3 same as -2 + elemental forces

=-4 same as -3 + [B] for 1st 2 Gauss pts. of the element

=-5 print [D] matrix only for 1st 2 Gauss pts. of the element

=-6 print [B] matrix only for 1st 2 Gauss pts. of the element

=-10 print elemental nonlinear stiffness

<-10 ICAN material property print when mat'l props generated through ICAN

=-11 same as -1 except individual layers not printed

=-13 print elemental forces only

**KSAVE** element stiffness save/read flag

=0 ignore

=1 read element stiffnesses from a file named FILE7 and integration point info from a file named FILE22 for 1st load case

**NOTE:** KSAVE option will not work properly for thermal loads or large displacement runs.

**LYPRNT** flag for layer orientation print to FILE18, printed as nodal displacement where node number is assigned from element input order beginning at NNMAX+1.

=0 ignore

=n print orientation at integration point n for each element layer

Enter element definition lines V.2. – V.5. only if (INGEOM=0)

V.2. 8 Noded Solid (NTYPEL = 8)

Line NEL N1 N2 N3 N4 N5 N6 N7 N8 IMAT

NEL Element number

N1—N8 Nodes defining the element, labeled as in Figure 3., repeated below

IMAT Material number (–1 if element removed)

V.3. 16 Noded Solid (NTYPEL = 16)

Line NEL N1 N2 N3 N4 N5 N6 N7 N8

Line N9 N10 N11 N12 N13 N14 N15 N16 IMAT

NEL Element number

N1—N16 Nodes defining the element, labeled as in Figure 3., repeated below

IMAT Material number (–1 if element removed)

V.4. 20 Noded Solid (NTYPEL = 20)

Line NEL N1 N2 N3 N4 N5 N6 N7 N8

Line N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 IMAT

NEL Element number

N1—N20 Nodes defining the element, labeled as in Figure 3., repeated below

IMAT Material number (–1 if element removed)

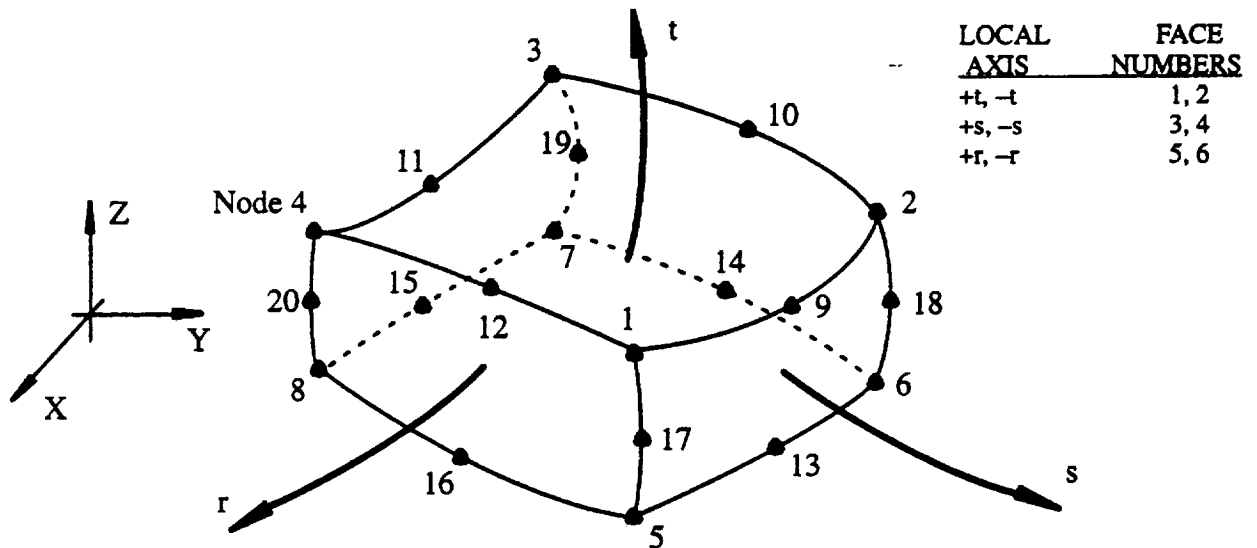


FIGURE 3. STRUCTURAL COORDINATE SYSTEM ORIENTATION FOR 3D ELEMENT

#### V.5. 4 Noded Shell (NTYPEL=4)

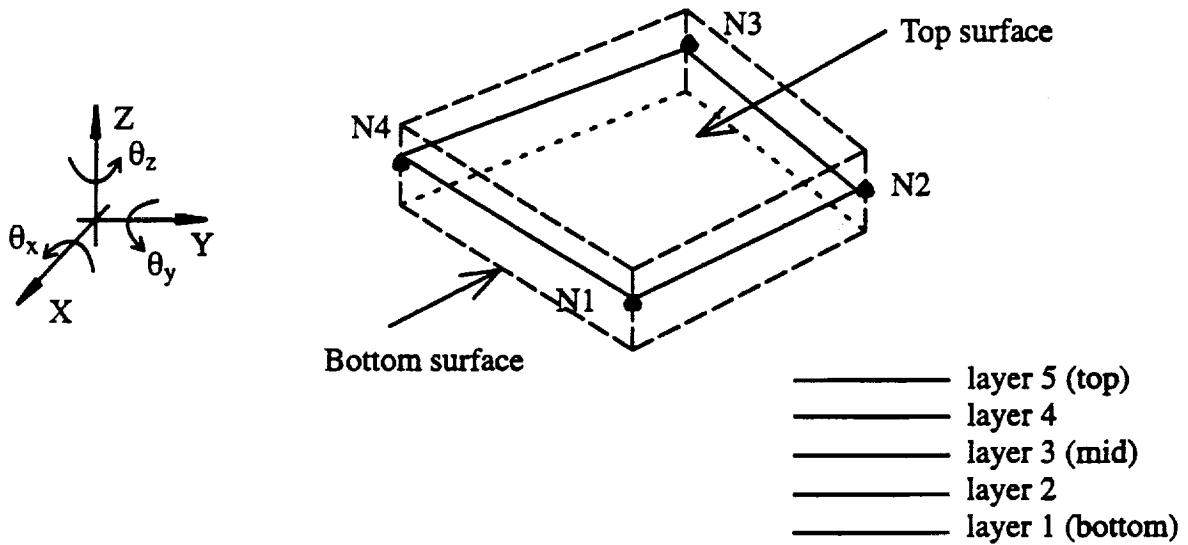
Line NEL N1 N2 N3 N4 IMAT THICK

NEL Element number

N1 – N4 Nodes defining the element

IMAT Material number

THICK Thickness [inch]



**NOTE:** This is the MHOST 4 noded shell element. It is currently isotropic only.

Enter mesh generation definition lines V.6. only if generating geometry ( $INGEOM < 0$ )

#### V.6. Mesh Generation Definitions

Enter V.6.a. – V.6.c.  $n$  times where  $INGEOM = -n$ .

##### V.6.a. Thickness Specification

Line **XI YI ZI XO YO ZO NELMT**

XI        = Inner surface center pt. global X coord.  
YI        = Inner surface center pt. global Y coord.  
ZI        = Inner surface center pt. global Z coord.  
XO        = Outer surface center pt. global X coord.  
YO        = Outer surface center pt. global Y coord.  
ZO        = Outer surface center pt. global Z coord.  
NELMT    = Number of elements through the thickness

**NOTE:** Thickness direction defined from inner to outer. Thickness vector must originate at global coordinate system origin and pass through (XI,YI,ZI) to (XO,YO,ZO) .

##### V.6.b. Rotation #1 Specification

Line **ANG1 RAD1 IAX1 NELM1**

ANG1     = Angle of rotation #1 [DEG.] (+ is CCW as viewed from positive end of IAX1)  
RAD1     = Radius of curvature of inner surface for rotation #1  
IAX1     = Local axis of rotation #1 (Euler axis)  
NELM1    = Number of elements in rotation #1 direction

##### V.6.c. Rotation #2 Specification

Line **ANG2 RAD2 IAX2 NELM2**

ANG2     = Angle of rotation #2 (DEG., + CCW)  
RAD2     = Radius of curvature for inner surface for rotation #2  
IAX2     = Local axis of rotation #2 (Euler axis)  
NELM2    = Number of elements in rotation #2 direction

**NOTE:** Euler rotations are automatically performed on global system until local X axis lies from (XI,YI,ZI) to (XO,YO,ZO). IAX1 and IAX2 are axes in the resulting coordinate system and cannot be the local X axis.



#### V.6.d. Layer Orientation Definition

Line **NGEN**

NGEN     Number of generation sets (Max of 25)

*Enter NGEN of the following line sets of V.6.e.*

#### V.6.e. Layer Material Definition

Line **NSET NLAY**

NSET     Generation set number

NLAY     Number of layers in the following set (Max of 10)

*Enter NLAY of the following line*

Line **MAT ANG THICK**

MAT     Material number of this layer

ANG     Orientation angle of principal direction with respect to the orientation axis [DEG.]  
(+ CCW as viewed from inner surface to outer)

THICK   Thickness of this layer

#### V.6.f. Generation Order

Line **IGEN(I), I=1,N**

IGEN     Array of generation numbers in order of layup starting from interior surface  
 $1 \leq N \leq 25$

**NOTE:** These inputs are used when generating the mesh internally. This internal generation is done such that the through the thickness direction (perpendicular to layers) is the element T axis; therefore, the T axis is the axis about which material properties are rotated. The material principle direction lies in the R-S plane with 0 degrees along the R axis. The R axis lies in the -Y'' direction. (See notes on mesh generation on following page.)

## NOTES ON MESH GENERATION INPUT (V.6.)

The mesh is generated from a local coordinate system which is formed from the global coordinate system. Two Euler rotations are performed to get the local X axis to pass through the inner and outer surface points. The first rotation is about the global Z axis and the second rotation is about the resulting Y' axis. Thus the local coordinate system referred to is the double prime (") system resulting from these two Euler rotations. One or both of these rotations may be zero depending on the input specification of the inner and outer surface points.

The resulting element connectivity is formed so that the element R axis is aligned with the direction, the element S axis is aligned with the  $-Z''$  direction, and the element T axis is aligned with the  $X''$  direction.

The input rotation specifications #1 and #2 are relative to the double prime system and the mesh is generated by sweeping the local X axis first about the specified IAX1 axis and then the IAX2 axis. Thus, in the general case, IAX1 will be either the  $Y''(2)$  axis or the  $Z''(3)$  axis and IAX2 will be either the  $Z''$  or  $Y''$  axis respectively (whichever was not IAX1). Some special geometry generation cases are listed below.

The number of nodes that will be created in a surface can be calculated based on the input specifications. The equations are listed on the following page. It is often prudent to check the generated geometry before embarking on a full scale analysis. This can be done by specifying the variable N described in Section I.2. as a value other than the actual number of generated nodes. A value of zero (0) will work well. This will cause execution to stop after the generated geometry is printed to the output echo file, NT1. This output can then be checked that the desired geometry was generated.

### GENERATION OF SPECIAL GEOMETRIES:

Cylinder: For either rotation #1 or #2, but not both

ANG = 0.0

RAD = length of cylinder along its axis

IAX = not used

The other rotation defines the cylinder curvature.

Cone: IAX1 = IAX2 and both ANG1 and ANG2  $\neq$  0.0

ANG1 = circumferential (curvature) angle

RAD1 = radius at origin end of cone

IAX1 = cone axis (=IAX2)

ANG2 = cone angle (angle with cone axis)

RAD2 = radius at far end of cone

IAX2 = cone axis (=IAX1)

Plane: both ANG1 and ANG2 = 0.0

RAD1 = length along IAX1

RAD2 = length along IAX2

### NUMBER OF NODES GENERATED

$$NN = FP + NELM1 * (FP + MP)$$

where

NELM1 is input

NELM2 is input

NELMT is input

$$FP = C * (NELM2 + 1) + M * NELM2$$

$$MP = M * (NELM2 + 1)$$

and for

8 node brick

$$C = NELMT + 1$$

$$M = 0$$

20 node brick

$$C = (2 * NELMT) + 1$$

$$M = NELMT + 1$$

*Enter VL only if INGEOM equals zero*

## VI. ELEMENT SKEW MATERIAL COORDINATE SYSTEMS

**NOTE:** These axes are used to define principal directions for orthotropic materials

Line **NESKEW**

NESKEW Number of element skew coordinate systems defined.

*Enter NESKEW of the following lines.*

Line **NEL NTRN NEND NINC**

NEL	Element number having a local material coordinate system
NTRN	Transformation number defining this skew
NEND	Last element having this local coordinate system
NINC	Increment on elements from NEL to NEND having this local coordinate system

Enter VII only if *INGEOM* equals zero

## VII. ELEMENT LAYER DEFINITION

Two types of layering input are available. The type described in Section VII.1. assigns layers to the elements directly, while that described in Section VII.2. assigns layers to a cross section of elements with the program determining the assignment of layers to the elements automatically.

In both types of layering the material orientations can be specified on a local or global level. On the local level, an angle about the through the thickness element axis is used. Angles are positive counter-clockwise as viewed from the positive end of the through the thickness axis. If the through the thickness axis is the *r* axis (*LAX*=1), the zero degree reference axis is the *s* axis. If the through the thickness axis is the *s* axis (*LAX*=2), the zero degree reference axis is the *t* axis. If the through the thickness axis is the *t* axis (*LAX*=3), the zero degree reference axis is the *r* axis. On the global level, a transformation number referring to a transformation of the global system is used. The transformations themselves are input in Section VIII.

An option is available that allows a combination of local and global specification of the material orientation. This option is activated using the variable *IGAX* together with the angular method of material orientation. The material system is formed based on the element local system and is then rotated about the through the thickness axis until the material 1 axis (zero degree reference) is perpendicular (*IGAX* > 0) or parallel (*IGAX* < 0) to the projection of the global axis specified by *IGAX* on the 1-2 material plane. The specified off axis angle is then measured from this orientation of the zero degree reference axis, which is related to global.

The layering definition can be used directly to generate layered elements, or it can be used to generate equivalent orthotropic material properties for use in homogeneous elements. Layered elements contain integration points at the midplane of each layer within the element. Homogeneous elements are composed of the same material throughout the element and are integrated with a Gaussian quadrature. The elements can be used together within a model. Assignment of this option occurs on the cross section level, so that elements through the thickness of the model must be of the same type (layered or homogeneous).

## VII.1. Manual Layer Input

### Line NELAY IGAX

NELAY    Number of manual layer definitions to be read  
          (may be zero, which causes program to go to VII.2.)

IGAX     Global axis to make perpendicular or parallel to element system reference axis  
          A two digit combination indicates radial and tangential axes. The tangential axis  
          will be made perpendicular or parallel to element system reference axis.  
          0=leave as is, >0 perpendicular, <0 parallel, where 1=X, 2=Y, 3=Z

**NOTE:** IGAX is read regardless of the value of NELAY.

*Enter NELAY of the following 2 line sets*

### Line NEL NLAYR LAX ISMEAR NEND NINC

NEL        Element for which this layer definition applies

NLAYR    Number of layers in this element

LAX        Local element system thickness (thru layer) axis.  
          (For models in which the mesh cannot be used to define the exact thru thickness definition, LAX is entered as a two digit code. The tens place defines the thru thickness axis, and the ones place defines which of the other two axes is to be used as a basis for the orthogonal material axis system. See Sections 2.4.3. and 4.1.4.4.)

ISMEAR   Bulk material property flag  
          =0 Use layered elements  
          =1 Use homogeneous elements with equivalent material properties

NEND      Ending element with this layer system

NINC      Increment in element number

*Enter NLAYR of the following line*

**NOTE:** Enter layers in the order as encountered when starting at negative end of LAX and moving the positive direction.

### Line IMAT PER ANG LTRAN

IMAT      Material number of this layer

PER        Decimal fraction of element thickness occupied by this layer

ANG        Angle (DEG.) rotated about element axis LAX, referencing orientation of material axes with respect to element local (used only if LTRAN=0)

LTRAN     Transformation number of skew system, referencing orientation of material axes with respect to global (if LTRAN=0, ANG is used; if LTRAN is negative, the transformation is from material to element local)

## VII.2. Cross Section Layup Generation

### VII.2.a. Layer Specification

#### Line **NGEN IGAX**

**NGEN**    Number of generation sets (may be zero, which causes program to go to VIII.)  
**IGAX**    Global axis to make perpendicular or parallel to element system reference axis  
            A two digit combination indicates radial and tangential axes. The tangential axis  
            will be made perpendicular or parallel to element system reference axis.  
            0=leave as is, >0 perpendicular, <0 parallel, where 1=X, 2=Y, 3=Z

**NOTE:** IGAX is read only if NGEN is not zero.

*Enter NGEN of the following two line sets*

#### Line **NSET NLAY**

**NSET**    Generation set number  
**NLAY**    Number of layers in this set

*Enter NLAY of the following line*

#### Line **LMAT THK ANG LTRAN**

**LMAT**    Material number for this layer  
**THK**    Layer thickness [INCHES]  
**ANG**    Orientation angle about thickness axis, [DEG] referencing material axes to element  
            local (used only if LTRAN=0)  
**LTRAN**    Transformation number of skew system, referencing orientation of material axes  
            with respect to global (if LTRAN=0, ANG is used; if LTRAN is negative, the trans-  
            formation is from material to element local)

#### Line **NORDER**

**NORDER**    Number of generation orders to be input

#### Line **IGEN(1) IGEN(2) ... IGEN(n)**

**IGEN(n)**    Generation set numbers used in layup. ( $1 \leq n \leq 24$ )  
            The order will be repeated from the beginning if more layers are needed to fill the  
            cross section thickness. If a set number is entered as a negative in IGEN the repeated  
            order will begin with that set.

## VII.2.b. Layup Generation

### Line **NLAYUP**

**NLAYUP** number of layup sequences to be generated.

0 indicates all cross sections will have layups generated individually as prescribed VII.2.c.

*Enter NLAYUP of the following line*

### Line **L1 LAX IORDR (LEND)**

**L1** First element in layup cross section

**LAX** Element axis aligned with layup thickness

(The input layup is assumed to be 1/2 of a symmetric layup. Negative **LAX** signals that the entire layup is input.)

(For models in which the mesh cannot be used to define the exact thru thickness direction, **LAX** is entered as a two digit code. The tens place defines the thru thickness axis, and the ones place defines which of the other two axes is to be used as a basis for the orthogonal material axis system. See Sections 2.4.3. and 4.1.4.4.)

**IORDR** Generation order to be used with this cross section

**LEND** Optional end of cross section. Used if cross section does not end at surface.

(If **LEND** is specified **L1** does not have to be on the surface, in which case it is assumed to be at the negative **LAX** end of the cross section.)

**NOTE:** Symmetric layups are generated such that the first layer in the first generation set begins at cross section outer surface. Layer generation then proceeds toward the midsurface. Fracture thicknesses are filled with the current layer.



### VII.2.c. Cross Section Specification

**NOTE:** This section can be used in one of two ways. It can be used in conjunction with the previous section (VII.2.b.) to assign the generated layup to several cross sections having the same thickness and number of elements through the cross section. This section can alternatively be used to generate layups for each specified cross section.

#### Line NCROSS

NCROSS Number of cross sections to be filled with a layup

*Enter NCROSS of the following line*

#### Line LMBEG LAX IORDR ISMEAR LXEND LXINC (LMEND)

LMBEG First element in the cross section

LAX Element axis aligned with cross section thickness

(All elements in the cross section must have the same thickness axis. The thickness axis for this particular cross section does not have to be the same as that used in VII.2.b. to determine the element layering.)

(When generating layups, the input layup is assumed to be 1/2 of a symmetric layup. Negative LAX signals that the entire layup is input.)

(For models in which the mesh cannot be used to define the exact thru thickness direction, LAX is entered as a two digit code. The tens place defines the thru thickness axis, and the ones place defines which of the other two axes is to be used as a basis to form the orthogonal material axis system. See Sections 2.4.3. and 4.1.4.4.)

IORDR Generation order used when generating layups, layup number when assigning layups to cross sections

ISMEAR Bulk material property flag

=0 Use layered elements in cross section

=1 Use homogeneous elements with equivalent material properties

LXEND First element in ending cross section

LXINC Increment in element number between cross sections. Corresponding elements of a cross section must increment the same

LMEND Optional end of cross section. Used if cross section does not end on surface.

**NOTE:** LMBEG must be on a surface unless LMEND is specified. In this case LMBEG is assumed to be at the negative LAX end of the cross section, which will go from LMBEG to LMEND or a surface element, whichever is encountered first.

## VIII. SKEW COORDINATE SYSTEMS

### VIII.1. Reference Nodes

Line **NSKEWR NREFNO**

NSKEWR Number of skew coordinate systems defined with reference nodes.

NREFNO Number of reference nodes that will be input

*Enter NREFNO of the following line*

Line **NOREF X Y Z**

NOREF Reference node number

(Must be different than structural node number,  
but not limited by max structural node name)

X,Y,Z Reference node global coordinates.

*Enter NSKEWR of the following lines.*

Line **NTRN NO NI NJ**

NTRN Transformation number associated with this skew system  
(NTRN < 0 will print transformation matrix on file NTDBG)

NO Node on local x axis

NI Node on x axis in +x direction

NJ Node in xy plane in general direction of +y axis

### VIII.2. Direction Cosines

Line **NSKEWD**

NSKEWD Number of skew coordinate systems defined with direction cosines.

*Enter NSKEWD of the following line*

Line **NTRN A11 A12 A13 A21 A22 A23**

NTRN Transformation number associated with this skew system

(NTRN < 0 will print transformation matrix on file NTDBG)

A11,A12,A13 Direction cosines defining local x axis.

The local x axis is defined by  $e_x = A11*i + A12*j + A13*k$

A21,A22,A23 Direction cosines defining local y axis.

The local y axis is defined by  $e_y = A21*i + A22*j + A23*k$ .

The local z axis is in the direction that forms a right handed coordinate system.

### VIII.3. Euler Angles

Line **NSKEWE**

NSKEWE Number of skew coordinate systems defined with Euler angles.

*Enter NSKEWE of the following line*

Line **NTRN ALPHA BETA GAMMA**

NTRN Transformation number associated with this skew system

(NTRN < 0 will print transformation matrix on file NTDBG)

ALPHA Right hand Euler rotation angle about the global X axis (DEG.)

BETA Right hand Euler rotation angle about the local y' axis (DEG.)

GAMMA Right hand Euler rotation angle about the local z'' axis (DEG.)

## IX. FIXED BOUNDARY CONDITIONS

**NOTE:** The specified degrees of freedom associated with these nodes and displacements will be eliminated from the set of equations. If a local coordinate system has been applied to a node using Section III., the displacement degrees of freedom are with respect to the local system. All degrees of freedom may not be active for a given element type. For example, solid isoparametric elements have only the 3 translational degrees of freedom active. Nodes which are not attached to any element must be fixed in all 6 degrees of freedom.

Line **NFIXBD**

**NFIXBD** Number of fixed boundary condition lines.

*Enter NFIXBD of the following lines*

Line **N IDX IDY IDZ IDTX IDTY IDTZ NEND NINC**

<b>N</b>	Node number
<b>IDX</b>	x translation flag = 0 unspecified or free x displacement = 1 fixed ( 0.0 ) x displacement to be eliminated
<b>IDY</b>	y translation flag = 0 unspecified or free y displacement = 1 fixed ( 0.0 ) x displacement to be eliminated
<b>IDZ</b>	z translation flag = 0 unspecified or free z displacement = 1 fixed ( 0.0 ) x displacement to be eliminated
<b>IDTX</b>	x rotation flag = 0 unspecified or free rotation about the x axis = 1 fixed ( 0.0 ) rotation to be eliminated
<b>IDTY</b>	y rotation flag = 0 unspecified or free rotation about the y axis = 1 fixed ( 0.0 ) rotation to be eliminated
<b>IDTZ</b>	z rotation flag = 0 unspecified or free rotation about the z axis = 1 fixed ( 0.0 ) rotation to be eliminated
<b>NEND</b>	Last node having this fixity condition
<b>NINC</b>	Increment on nodes from N to NEND having this fixity

## X. STIFFNESS ADDITIONS

Enter KADD of the following lines.

Line NOD1 NOD2 NDOF M1 M2

- NOD1 First node to which these additional stiffness terms apply.  
This node must have been defined in the nodal input.
- NOD2 Second node to which this stiffness applies.  
This node must have been defined in the nodal input.
- NDOF Number of degrees of freedom being related by the additional stiffness terms.  
( $1 \leq \text{NDOF} \leq 6$ )
- M1 Mass input flag for NOD1.  
=0 No masses input for NOD1; =1 Read mass input for NOD1
- M2 Mass input flag for NOD2.  
=0 No masses input for NOD2; =1 Read mass input for NOD2

Line KA(1,1) ..... KA(1,NDOF\*2)

: :  
:  
Line KA(NDOF\*2,1) ... KA(NDOF\*2,NDOF\*2)

	U1	V1	W1	U2	V2	W2
U1	k	0	0	-k	0	0
V1	0	k	0	0	-k	0
W1	0	0	k	0	0	-k
U2	-k	0	0	k	0	0
V2	0	-k	0	0	k	0
W2	0	0	-k	0	0	k

KA(n,n) Stiffness terms to be added into structural stiffness matrix. The terms are entered as a square matrix which is 2\*NDOF by 2\*NDOF square as shown above (in the box) for a typical linear ground spring type of application for the translational DOF.

**NOTE:** The additional stiffness terms are input as a square matrix with degrees of freedom activated in the order of X,Y,Z,RX,RY,RZ; i.e. even if only the RX (X rotation) terms are to be related NDOF=4 must be entered and an 8x8 KA adder matrix would be entered with most terms = 0. All degrees of freedom may not be active for a given element type. For example, nodes of solid isoparametric elements have only the 3 translational degrees of freedom active, requiring a 6x6 matrix as shown schematically above. Nodes need not be connected to any other elements, in which case all 6 dof may be active as defined by the nodal fixities of Section IX.

Enter the following line only if M1  $\neq$  0

Line MASS1(1) ..... MASS1(NDOF)

Enter the following line only if M2  $\neq$  0

Line MASS2(1) ..... MASS2(NDOF)

MASS Lumped mass terms for each degree of freedom input.

**NOTE:** Translation dof normally have the same mass value. Rotational dof are rotary inertias, J.

## XI. LOAD CASE CONTROL INFORMATION

### XI.1. Load Case Control Cards

Line **RPM IAXS IACC IELA IPC IEIG IHT IBUCK IABSO ICAN IFC MOSAIC**

**RPM** Rotational speed ( RPM )  
**IAXS** Global axis about which the structure is rotating  
1 = X, 2 = Y, 3 = Z, -1 = no RPM loads in analysis (timesaving)  
(For plane elements, rotation is about the X or Z axes only. For axisymmetric elements, rotation is always about the X axis.)  
**IACC** Indicator for input of acceleration loads  
0 = no, 1 = yes, -1 = never in analysis (timesaving)  
**IELA** Indicator for a change in elastic properties  
0 = no changes  
1 = change elastic material properties as defined in Section XIV.1, or XIV.2.  
-1 = change elastic properties as calculated using ICAN module (Section XIV.3.)  
2 = change user material properties as defined in Section XIV.5.  
12 = change elastic material properties and user material properties

**NOTE:** If IELA=0 is indicated in the first load case, material properties will be expected as described in Section XIV.1, or XIV.2.

**IPC** Indicator for a change in plastic/creep/damage properties  
0 = no changes  
1 = change plastic properties  
2 = change creep properties  
3 = change damage properties  
combinations of these codes are allowed (i.e. 12, 13, 123)  
4 = change plasticity controls (<0 debug on)  
**IEIG** Eigen analysis flag (also secondary acoustic flag)  
0 = No eigenvalue/eigenvector analysis  
<0 The IEIG lowest eigenvalues/vectors will be computed using the determinant search technique  
>0 Eigenvalue/eigenvector analysis using subspace iteration technique

**NOTE:** Mode shape slope calculations can be performed based on input from the file SLOP, which contains variables NLE, NMID, NTE, IAXI, and ITAN. See Chapter 4.1.1.4.

- IHT** Heat transfer load case indicator
- 0 = No heat transfer analysis
  - 1 = Linear steady state
  - 2 = Nonlinear steady state
  - 3 = Linear transient
  - 4 = Nonlinear transient
  - <0 = continuation of previous heat transfer load case (no additional heat transfer input read)
- IBUCK** Buckling analysis indicator
- 0 = No buckling analysis
  - >0 solve for lowest IBUCK load multipliers using original configuration stiffnesses
  - <0 solve for lowest IBUCK load multipliers using current configuration stiffnesses
- IABSO** Absorptivity analysis indicator
- 0 = No absorptivity analysis
  - 1 = Absorptivity analysis is performed
- ICAN** Integrated composites analyzer indicator

**NOTE:** Material properties must be generated by the ICAN module (IELA = -1) to perform a full ICAN analysis.

- 0 = No ICAN analysis
- 1 = full ICAN analysis using displacements as input
- 2 = full ICAN analysis using calculated loads as input
- 3 = cross section interpolations only (material axes)  
(negative to print output in UOF form)
- 4 = cross section interpolations only (global axes, Z through thickness)  
(negative to print output in UOF form)
- 5 = cross section interpolations only (material zero degree orientation)  
(negative to print output in UOF form)
- 1 = full ICAN analysis using interpolated ply strains
- 10 = full ICAN analysis using interpolated ply strains and stresses.

- IFC** Failure criteria indicator

**NOTE:** Failure criteria are calculated using stresses (or strains) oriented as defined by the variable IPOUT.

- 0 = no failure criteria calculations
  - 1 = do failure criteria calculations
- MOSAIC** Output flag for MOSAIC submodels
- 0 = no MOSAIC output
  - 1 = Write output for MOSAIC submodels (NODE, X,Y,Z,  $\delta X, \delta Y, \delta Z$ ) to FILE12

Line NDIS NFOR NTEM NPRE NADD IPOUT TIME NTIMND NFAIL IEIGST NEOUT

NDIS      Number of nodal displacement inputs  
NFOR      Number of nodal force inputs  
NTEM      Number of temperature inputs  
          -1 = no thermal loads in analysis (timesaving)

**NOTE:** For first load case nodal temperatures are initialized at 0.0 unless otherwise specified.

NPRE      Number of pressure loads input  
NADD      Number of element addition/removal inputs  
IPOUT      Printout options flag  
          =0 all results are printed for this load case  
          =-1 no printout for this load case

Partial printout is obtained using a 4 digit packed code:

Ones location controls displacements, 1=print, 2=print in E, >2=summary only, 0=no

Tens location controls reactions, 1=print, >1=summary only, 0=no print

Hundreds location controls detailed stress/strain printout, 0=no print

1=print in global (ksi), 2=print in material local (ksi), 3=print principles (ksi),

4=print in global (psi), 5=print in material local (psi), 6=print principles (psi),

8=print in material local (psi) with temperatures and no banners (mission form),

9=print in-plane principles and max interlaminar (ksi)

Thousands location controls max/min stress/strain printout

If detailed stress/strain is on this is automatically set to the same value, otherwise

0=don't print, 1=print in global, 2=print in material local

TIME      Time elapsed in this load case

NTIMND    Heat transfer time point where structural solution is desired;  
          used with transient heat transfer only

NFAIL      Number of failed stiffness lines to be read  
          -1 = no failed stiffnesses in analysis (timesaving)

IEIGST    Mode shape stress flag  
          Output is printed as indicated by IPOUT (orientation and units)  
          Negative values indicate output is to be in UOF form  
          Enhanced interlaminar calculations used. Turn off by 1 in hundreds location.

=0 No mode shape stresses calculated

=1 Calculate mode shape stresses for top and bottom layers in each element

=10 Calculate mode shape stresses for all layers in each element

>10 Calculate mode shape stresses for all layers for modes (IEIGST-10)

NEOUT    Selective element output flag

=0 print stress/strain results for all elements

≠0 print stress/strain results for first NEOUT elements listed on input deck only



*XII. is interpreted only if the keyword string is the first entry on the line (excluding a line number)*

## XII. KEYWORD INPUT

The occurrence of the keyword string causes the subsequent entries to be interpreted as the input associated with the keyword. The keyword string must be capitalized and separated by at least one space from any other entries. Keyword input may be in any order.

### XII.1. Eigenanalysis Solution Parameter Keyword: EIGS

Line **EIGS SC RTOL NITEMP NSPACE IFPR SHIFT**

**EIGS** Character string EIGS

**SC** Scale factor multiplier.

A scale factor is calculated and applied to the mass terms to reduce the order of magnitude difference between the mass and stiffness terms. The calculated scale factor is multiplied by SC. A value of zero input for SC is interpreted as the default value of 1.E-5.

**RTOL** Convergence tolerance on subsequent iteration eigenvalues.

A value of zero input for RTOL is interpreted as the default value of 1.E-5.

**NITEMP** Max number of iterations allowed.

A value of zero for NITEMP is interpreted as the default value of 20.

**NSPACE** Number of extra subspace vectors.

The vector subspace used is either twice the requested number of eigenvalues or the requested number of eigenvalues + NSPACE, whichever is smaller. A value of zero input for NSPACE is interpreted as the default value of 8.

**IFPR** Extra print flag.

This controls the amount of detailed eigenanalysis information printed out on FILE66. A value of zero for IFPR is interpreted as the default value of 1.

**SHIFT** Eigenvalue shift value

This value is applied to eigenvalues during solution and is later subtracted from the calculated eigenvalues.

### XII.2. Alternate Stress Stiffening Keyword: RDKS

Line **RDKS IKSTR**

**RDKS** Character string RDKS

**IKSTR** Alternate Stress Stiffness Flag

=0 generate stress stiffness using current stresses and solve (normal buckling)

=1 generate stress stiffness only

=2 read stress stiffness from a file (previously generated FILE7)

### XII.3. Large Displacement Buckling Keyword: LDBK

Line **LDBK LDBCK**

LDBK Character string LDBK

LDBCK Large Displacement Buckling Flag

=0 Stress Stiffness included in Large Displacement analysis (i.e. Off)

=1 Stress Stiffness no included in Large Displacement analysis (i.e. On)

LDBCK retains its value until explicitly changed in a subsequent load case

### XII.4. Plasticity Debug Print Keyword: PDBG

Line **PDBG IDBGP**

PDBG Character string PDBG

IDBGP Plasticity debug flag

=0 no debug output print

≠0 value is element for which debug output is printed

### XII.5. Damage Debug Print Keyword: DDBG

Line **DDBG IDBGD**

DDBG Character string DDBG

IDBGD Damage debug flag

=0 no debug output print

≠0 value is element for which debug output is printed

### XII.6. Damage Acceleration Keyword: DACC

Line **DACC NAITK**

DACC Character string DACC

NAITK Aitken acceleration iteration interval

=0 no acceleration (must be explicitly set to 0, blank NAITK interpreted as default)

≠0 value is iteration interval for acceleration (default = 3)

DACC only applicable for right hand side (IDMO=1) pseudo force (METHOD=2)

### XII.7. Damage Iteration Averaging Keyword: DAVG

Line **DAVG IAVG**

DAVG Character string DAVG

IAVG Damage iteration averaging flag

=0 no averaging of elastic matrix [D] or damage strain  $\{\epsilon\}_d$

=1 average [D], not  $\{\epsilon\}_d$ ; =2 average  $\{\epsilon\}_d$ , not [D]; =3 average both [D] and  $\{\epsilon\}_d$

#### XII.8. Creep Debug Print Keyword: CDBG

Line **CDBG IDBGC**

CDBG Character string CDBG

IDBGC Creep debug flag

=0 no debug output print

≠0 value is element for which debug output is printed

#### XII.9. Vibration BC Change Keyword: VIBC

Line **VIBC IVBCHG**

VIBC Character string VIBC

IVBCHG Vibration BC Change Flag

Number of prescribed displacement lines to be read prior to frequency calculation.

Stiffness is modified to reflect these changes so that BC for vib solution is different than BC for static solution. The prescribed displacement input lines should be placed immediately before the ICAN ANALYSIS INFORMATION input of Section XXV.

#### XII.10. Newton–Raphson Keyword: NRIT

Line **NRIT NRITER**

NRIT Character string NRIT

NRITER Newton–Raphson nonlinear iteration flag (applies only for IPLO < 0)

=0 modified Newton–Raphson (stiffness unchanged through iteration process)

=1 full Newton–Raphson (stiffness changed after each iteration)

>1 modified Newton–Raphson (stiffness changed after each NRITER'th iteration)

NRITER retains its value until explicitly changed in a subsequent load case

#### XII.11. Nonlinear Force Calculation Keyword: METH

Line **METH METHOD**

METH Character string METH

METHOD Nonlinear force calculation flag

=0 use default (=2, pseudo force)

=1 calculate force as residual (applied – internal)

=2 calculate force as pseudo (directly from nonlinear strains)

This keyword is applicable for initial stiffness (right hand side) techniques only

METHOD retains its value until explicitly changed in a subsequent load case

#### XII.12. Microstress Plot Keyword: MICR

Line **MICR MICRO**

**MICR** Character string MICR

**MICRO** Microstress plot flag

=0 No output for microstress plots

=1 Create output for microstress plots on version numbered files PATMICROF, PATMICROA and PATMICROB for fiber, matrix region A, and matrix region B

Load case control variable ICAN must specify a full ICAN analysis (1,2,-1, or -10) and the analysis locations should be specified in Section XXV, usually as NCS = -

#### XII.13. Material Selected Stress Summary Keyword: MATX

Line **MATX MATS(1) ... MATS(JMAT)**

**MATX** Character string MATX

**MATS** Vector of materials selected for stress summary

Up to JMAT materials may be specified. Currently JMAT=3. If less than JMAT materials are specified, the overall summary will include all materials. If JMAT materials are specified, the overall summary will include only the JMAT materials.

MATS retains its values until explicitly changed in a subsequent load case

#### XII.14. Load Subincrementing Keyword: RAMP

Line **RAMP NSUB**

**RAMP** Character string RAMP

**NSUB** Number of load subincrements

Boundary conditions for the current load case will be applied in a linearly ramped fashion over the number of specified load subincrements. Temperatures are not ramped so that stiffnesses corresponds to the final temperature for the load case throughout load subincrementing. However, thermal loads are ramped along with applied loads (RPM, pressure, accel, prescribed displacement).

NSUB=0 turns off load subincrementing and is essentially equivalent to NSUB=1.

## XII.15. Cross Section Layer Output Keyword: XSEC

### Line **XSEC IXPAT**

**XSEC**     Character string XSEC

**IXPAT**     PATELSIG cross section layer output flag.

          =0 PATELSIG results written for element layers

          =1 PATELSIG results written for cross section layers

For layered elements, PATRAN results file PATELSIG contains stress results at centroid of each layer. Normally these layers are output for element layers: each element has a layer 1 and layers are output from 1 to the number of layers in the element.

Since each element record must contain the same amount of information, each element record contains layer stress records up to the maximum number of layers in any element in the model. For layer records greater than the number of layers in a specific element, all stress components are written = 0.

Setting IXPAT=1 will write layers by cross section number, and any cross section layer not within a specific element is written with all stress components = 0. Cross section layer numbers are determined and stored during the cross section layering process (see Section VII.2.) so this option is not usable with manual layering. Note that the number of layers in an element record will be the maximum number of layers in any cross section when IXPAT=1.

Enter XIII only if IACC = 1

**XIII. ACCELERATION SPECIFICATION FOR INERTIA AND GRAVITY LOADS**

Line **ACCELX ACCELY ACCELZ**

ACCELX Acceleration of the structure in the global X direction.

Units are [in/sec<sup>2</sup>]

ACCELY Acceleration of the structure in the global Y direction.

Units are [in/sec<sup>2</sup>]

ACCELZ Acceleration of the structure in the global Z direction.

Units are [in/sec<sup>2</sup>]

*Enter only if not using ICAN to generate properties*

#### XIV. MATERIAL ELASTIC PROPERTIES

*Enter MMAT groups of material physical properties*

Line MTN RTEM MTP DEN

MTN	Material number ( $1 \leq \text{MTN} \leq \text{MMAT}$ )
RETM	Stress free reference temperature for this material
MTP	Number of temperatures at which properties will be entered
DEN	Weight density of the material ( $\text{lb}_f / \text{in.}^3$ )

(CSTEM divides DEN internally by  $386 \text{ in./sec}^2$  to obtain mass density)

##### XIV.1. Isotropic Material

*Enter MTP lines if the material is isotropic*

Line TEMP E PR AL

TEMP	Temperature ( degrees F. )
E	Elastic modulus ( $10^6 \text{ psi}$ )
PR	Poisson's ratio
AL	Mean coefficient of thermal expansion ( $10^{-6} \text{ in./in./degF.}$ )

#### XIV.2. Orthotropic Material

*Enter MTP lines if the material is orthotropic*

Line **TEMP EX EY EZ PXY PYZ PXZ GXY GZY GXZ ALX ALY ALZ**

TEMP	Temperature ( degrees F. )
EX	Elastic modulus in the X direction ( 10**6 psi )
EY	Elastic modulus in the Y direction ( 10**6 psi )
EZ	Elastic modulus in the Z direction ( 10**6 psi )
PXY	Poisson's ratio relating X and Y strains
PYZ	Poisson's ratio relating Y and Z strains
PXZ	Poisson's ratio relating X and Z strains
GXY	Shear modulus in the X Y plane ( 10**6 psi )
GZY	Shear modulus in the Z Y plane ( 10**6 psi )
GXZ	Shear modulus in the X Z plane ( 10**6 psi )
ALX	Mean coefficient of thermal expansion in X direction (10**-6 in./in./deg. F.)
ALY	Mean coefficient of thermal expansion in Y direction (10**-6 in./in./deg. F.)
ALZ	Mean coefficient of thermal expansion in Z direction (10**-6 in./in./deg. F.)

**NOTE:** Poisson's ratios are related according to the equation

$$\frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j}$$



*Enter only if using ICAN routines to generate elastic material properties*

### XIV.3. ICAN Generated Elastic Properties

Line **NMS**

NMS      Number of different material systems

*Enter NMS of the following line*

Line **C1 C2 VFP VVP C3 C4 VSC VFS VVS TCU DELM NWEAVE**

C1      Primary fiber type data bank code  
C2      Primary matrix type data bank code  
VFP      Primary fiber volume ratio ( $0 < VFP < 1$ )  
VVP      Primary void volume ratio  
C3      Secondary fiber type data bank code  
          (same as C1 for standard composite systems)  
C4      Secondary matrix type data bank code  
          (same as C2 for standard composite systems)  
VSC      Secondary composite system volume ratio  
          (=0 for standard composite systems)  
VFS      Secondary fiber volume ratio  
VVS      Secondary void volume ratio  
TCU      Cure temperature (used as reference temperature)  
DELM      Percentage of moisture  
NWEAVE    Number of plies to combine in order to produce simulated weave properties  
          (=0 for unidirectional plies)

**NOTE:** Material numbers are assigned consecutively in order of input, i.e. first material system input is material number 1, second material system input is material number 2, etc.

**NOTE:** A separator (blank or comma) is required between C1 and C2 and between C3 and C4

*Enter NWEAVE of the following line*

Line **ANG THICK**

ANG      Orientation angle of the ply [deg]  
THICK      Ply thickness [inch]

**NOTE:** Simulated weave properties are obtained by calculating laminate properties for the combination of plies given above. All layers having a material number corresponding to this ICAN material specification will use these simulated weave properties.

Enter XIV.4, only if IFC is non-zero

#### XIV.4. Failure Criteria Strengths and Parameters

##### Line IMAT IFAIL

IMAT      Material number to which the following strengths and parameters apply  
            0 = end of this input (values from previous load case are reused if not input)

IFAIL      Failure criteria to be used for this material  
            1 = Tsai-Wu quadratic failure criteria  
            2 = Max Stress failure criteria  
            3 = Max Strain failure criteria  
            4 = Tsai-Hill failure criteria

##### Line FT XT XC YT YC ZT ZC

FT          Temperature at which these strengths apply [F]  
XT          Tensile strength in material X (1) direction [psi]  
XC          Compressive strength in material X (1) direction [psi]  
YT          Tensile strength in material Y (2) direction [psi]  
YC          Compressive strength in material Y (2) direction [psi]  
ZT          Tensile strength in material Z (3) direction [psi]  
ZC          Compressive strength in material Z (3) direction [psi]

##### Line SXY SYZ SXZ (FXY FYZ FXZ)

SXY          Shear strength in material XY (12) plane [psi]  
SYZ          Shear strength in material YZ (23) plane [psi]  
SXZ          Shear strength in material XZ (13) plane [psi]

The following parameters apply only when using the Tsai-Wu failure criteria (IFAIL=1)

FXY          Interaction parameter for material X and Y directions  
FYZ          Interaction parameter for material Y and Z directions  
FXZ          Interaction parameter for material X and Z directions

**NOTE:** A non-zero IFC means that failure criteria parameter input is read until a zero material number (IMAT=0) is encountered. Failure criteria will only be calculated for those materials that have failure parameters defined. For subsequent load cases, a single IMAT=0 line would indicate that previously input parameters are to be used in failure calculations. For use with IC strengths are calculated so simply enter a single IMAT=0 line. The max stress criteria is the only one available with ICAN calculated strengths and is automatically selected.

**NOTE:** Failure criteria analysis performed using stress/strain oriented in global or material as specified by IPOUT. If principle directions requested, failure criteria performed using global values.

Enter XIV.5. only if IELA=2, 12, or 21

XIV.5. must be entered if using User Routines for plasticity, creep, or damage

#### XIV.5. User Material Properties and Constitutive Data

Line **NMAT MQMX NIPS NRPS NEPS NSTATP NSTATM**

- NMAT    Number of materials for which properties are to be read  
         NMAT  $\neq 0$  if user material property file is to be used
- MQMX    Maximum number of values for any one material  
         (largest MATQ that will be input ; used to size storage record length)
- NIPS    Number of user storage locations needed in CHOOK integer property vector  
         (NIPS  $\geq 10$ )
- NRPS    Number of user storage locations needed in CHOOK real property vector  
         (NRPS  $\geq 15$ )
- NEPS    Number of user storage locations needed in CHOOK strain storage vector  
         (NEPS  $\geq 18$ )
- NSTATP   Number of user storage locations needed in CHOOK ply level state vector  
         (NSTATP  $\geq 30$ )
- NSTATM   Number of user storage locations needed in CHOOK micro level state vector  
         (NSTATM  $\geq 48$  if using a micro level model, NSTATM=0 if not)

Enter NMAT of the following two input line sets

Line **MTN MATQ**

- MTN    Material number (  $1 \leq \text{MTN} \leq \text{MMAT}$  )
- MATQ    Number of input values to be read for material MTN (  $0 \leq \text{MATQ} \leq \text{MQMX}$  )

Enter as many lines as needed to input the MATQ material values

Line **V(i)**

- V(i)    Material values needed for user constitutive model  
         i=1,MATQ (Max of 25 values per line, max line length of 80)

*Enter only if IPC contains a 1 (1, 12, 21, 123 etc.)*

## **XV. PLASTICITY DATA**

### **XV.1. Plasticity and Iteration Limit**

**Line NPMAT MPIT**

**NPMAT** Number of materials for which plasticity data is to be read  
-1 = No plasticity (turns off plasticity analysis for the current load case)  
0 = No plasticity data to be read (allows only MPIT to be changed)  
N = Plasticity data for N materials to be read

**MPIT** Maximum number of plasticity iterations to be performed.  
If the solution does not converge after MPIT iterations the program will stop.  
If the user wishes to continue with the next load case after non-convergence,  
input MPIT as a negative number.

*Enter NPMAT of Section XV.2. with the input of Sections XV.3. or XV.4. as appropriate*

### **XV.2. Plasticity Options and Convergence Tolerance**

**Line IMAT IPLAS NPTEM NPTS PTOL PCUTF PHARD**

**IMAT** Material number for which this plasticity data applies

**IPLAS** Plasticity option  
0 = isotropic hardening  
1 = subvolume kinematic hardening  
3 = CT Sun orthotropic plasticity  
10 = CHOOK user plasticity model  
User plasticity data entered as user material properties (See Section XIV.5.)

**NPTEM** Number of temperatures for which plasticity data is to be entered  
=0 no plastic material data to be read, any previously read data is retained  
≠0 reads material data at NPTEM temps, all previously read data is overwritten

**NPTS** Number of points used to specify the material stress strain curve ( $2 < \text{NPTS} < 10$ )  
If  $\text{NPTS} \leq 0$ , only a linear elastic solution will be done.

**PTOL** Convergence tolerance value, in strain or percent depending on sign of PTOL  
 $\text{PTOL} > 0$  absolute change in plastic strain increment  $< \text{PTOL}$   
(  $1.E-5$  recommended, default),  
 $\text{PTOL} < 0$  percent change in plastic strain increment  $< \text{PTOL}$   
(  $-.05$  (5%) recommended )

**PCUTF** Plastic strain cutoff (default  $\text{PCUTF} = 1.E-6$ )  
Plastic strain less than PCUTF will be assumed to be elastic.

**PHARD** Plasticity hardening parameter (used only for  $\text{IPLAS}=1$ )  
 $0 \leq \text{PHARD} \leq 1$ , where 0 = kinematic hardening, 1 = isotropic hardening

Enter XV.3. only if IPLAS = 0 or IPLAS = 1

Enter NPTEM of Section XV.3. in increasing order from temperature 1 to NPTEM.

### XV.3. Classical Plasticity Models

Line TEMP

Line	STRESS(1)	STRAIN(1)
------	-----------	-----------

Line STRESS(2) STRAIN2)

:

:

Line STRESS(NPTS) STRAIN(NPTS)



*Enter only if ICRO  $\neq$  0*

**XVI. RATE DEPENDENT INELASTICITY TIME INCREMENT CONTROL**

**Line N2M TINIT ECMAX SIGMAX ERMAX DELMIN DELMUL**

**N2M** Number of equal time steps in the load case.

If a zero is entered the dynamic time incrementing algorithm will be activated and the remaining data on this line is needed.

**TINIT** Initial time step for the load case.

If a zero is input the initial time step for the first load case will be  $.01 * \text{TIME}$ .

If zero is input for subsequent load cases the initial time step will be half of the time step calculated at the end of the previous load case.

**ECMAX** Maximum effective inelastic strain increment per time step.

(default is .000200)

**SIGMAX** Maximum effective stress change per time step.

(default is 4000 PSI)

**ERMAX** Maximum allowable integration error per time step.

(default is .02)

**DELMIN** Minimum allowable time step.

(default is  $.01 * \text{TIME}$ )

**DELMUL** Max multiplier on increment from current time step to the next.

(default is 1.5)

*Enter only if IPC contains a 2 (2, 12, 21, 123, etc)*

## XVII. MATERIAL TIME DEPENDENT PROPERTIES

### XVII.1. Creep and Iteration Limit

Line NCMAT MCIT

NCMAT Number of materials for which creep data is to be read

-1 = No creep (turns off creep analysis for the current load case)

0 = No creep data to be read (allows only MCIT to be changed)

N = Creep data for N materials to be read

MCIT Maximum number of creep iterations to be performed in a time step.

If the solution does not converge after MCIT iterations the program will stop.

If the user wishes to continue with the next time step after non-convergence, input MCIT as a negative number.

*Enter NCMAT of Section XV.2, with the input of Sections XV.3, or XV.4, as appropriate*

### XVII.2. Creep Options and Convergence Tolerance

Line IMAT ICRP NCTEM IHARD SIGNRM CTOL TCUT SCUT

IMAT Material number for which this creep data applies

ICRP Creep option

0 or 1 = 5 term classical creep model

2 = CT Sun orthotropic creep model

10 = CRHOOK user creep model

NCTEM Number of temperatures for which creep data is to be entered

=0 no creep material data to be read, any previously read data is retained

≠0 reads material data at NCTEM temps, all previously read data is overwritten

IHARD creep hardening rule

0 = time hardening

1 = strain hardening

SIGNRM Normalization stress in psi, default is 100000.

CTOL Convergence tolerance value, in psi or decimal percent depending on sign of CT

CTOL>0 absolute change in effective stress increment < CTOL

( 100 psi recommended, default),

CTOL<0 percent change in effective stress increment < CTOL

( -.01 (1%) recommended )

TCUT Temperature (deg. F) below which creep will be ignored.

If the temperature is less than TCUT then the creep strain rate will be zero.

SCUT Stress (psi) below which creep will be ignored.

The SCUT option is not currently available.



Enter XVII.3. only if using 5 term classical creep law (ICRP = 1)

### XVII.3. Classical Creep Formulation

Enter *NCTEM* of Section XVII.3.a.

#### XVII.3.a. Creep Constants

Line **TEMP**

TEMP      Temperature at which creep properties are input

Line **K N M Q R**

K, N, M, Q, R      Constants in the creep equation:

$$\bar{\epsilon}_c = K \bar{\sigma}_t^{N,M} + Q \bar{\sigma}_t^R$$

where the normalized effective stress,  $\bar{\sigma} = \text{SIG}/\text{SIGNRM}$  and  $t$  is time in hours

*Creep rupture option is not currently available. Enter NRUPT = 0*

#### XVII.3.b. Creep Rupture Data

Line **NRUPT**

NRUPT      Number of temperatures at which creep rupture data is input  
Since creep rupture option is not currently available, enter 0

*Enter the following lines only if NRUPT  $\neq$  0*

Line **RUPTEM(1) RUPTEM(2) ... RUPTEM(NRUPT)**

RUPTEM( )      Temperatures at which creep rupture data is to be read

Line **STRESS(1,1) TIME(1,1)                      STRESS(1,2) TIME(1,2)**

Line **STRESS(NRUPT,1) TIME(NRUPT,1)      STRESS(NRUPT,2) TIME(NRUPT,2)**

**NOTE:** For each line the STRESS, TIME pairs specified define 2 points on the creep rupture curve at a given RUPTEM. Stresses are in psi and times are in hours.

Enter XVII.4. only if using C.T. Sun creep law (ICRP = 2)

Enter NCTEM of Section XVII.4.

XVII.4. C.T. Sun Orthotropic Creep Constants

Line TEMP

TEMP      Temperature at which creep properties are input

Line B C D m n r s

B C D m n r s      Constants in the C.T. Sun creep equation:

$$\bar{\epsilon}_c = B\bar{\sigma}^m t + C\bar{\sigma}^n (1 - \exp[-D\bar{\sigma}^r t])$$

where the normalized effective stress,  $\bar{\sigma} = \text{SIG}/\text{SIGNRM}$  and t is time in hours.

**NOTE:** The exponent n is the same as n in the power law eff stress – eff plastic strain curve, but must be input here even if entered for C.T. Sun orthotropic plasticity.

**NOTE:** The potential function parameters in Section XV.4.a. must be input as well.

If no plasticity is to be done for this material,

set NPTS < 0 to avoid input of stress–strain curves.

If no plasticity analysis at all is desired,

set IPLO = 0, IPC = 12 to read both plastic and creep properties, IPLAS = 3 to read CT Sun plasticity inputs, and NPTS < 0 to avoid input of stress–strain curves

*Enter only if IPC contains a 3 (3, 13, 23, 123 etc.)*

## **XVIII. DAMAGE DATA**

### **XVIII.1. Damage and Iteration Limit**

**Line NDMAT MDIT NCYC NCYPTS NCYCM IUNIT**

- NDMAT** Number of materials for which damage data is to be read  
-1 = No damage (turns off damage analysis for the current load case)  
0 = No damage data to be read (allows only MDIT, NCYC to be changed)  
N = Damage data for N materials to be read
- MDIT** Maximum number of damage iterations to be performed.  
If the solution does not converge after MDIT iterations the program will stop.  
If the user wishes to continue with the next load case after non-convergence,  
input MDIT as a negative number.
- NCYC** Number of mission cycles to be calculated for this loading condition  
(explicit load case cycles)
- NCYPTS** Number of cycle points (i.e. number of load cases comprising one cycle)  
The load condition describing each cycle point is a load case, beginning with the  
current load case, through NCYPTS load cases.  
If NCYPTS = 1, the cycle is simply from an unloaded condition to the load condi-  
tion of the current load case.  
If NCYPTS > 1, the cycle is from an unloaded condition to the load condition of the  
current load case, then through the next NCYPTS-1 load cases on the input deck.
- NCYCM** Number of minor cycles (response calculated by damage model)
- IUNIT** Units type flag. =0 for British units [psi, inch], =1 for SI units [MPa, meter]

Enter NDMAT of Section XVIII.2. with the input of Sections XVIII.3. or XVIII.4. as appropriate

XVIII.2. Damage Options and Convergence Tolerance Parameters

Line **IMAT MDAMG NDTEM DTOL ITYP**

**IMAT** Material number for which the following apply

**MDAMG** Damage model option

0 = no damage

1 = ADEAL (hardwired constants)

2 = Leckie CMCUMAT damage model

3 = VPI polynomial model

10 = DHOOK user damage model

**NDTEM** Number of temperatures for which damage data is to be entered

=0 no damage material data to be read, any previously read data is retained

≠0 reads material data at NDTEM temps, all previously read data is overwritten

**DTOL** Tolerance for material IMAT.

The convergence criteria used depends on the sign of TOL:

TOL>0 absolute change in convergence parameter < TOL

TOL<0 percent change in convergence parameter < TOL

**ITYP** Convergence type flag, indicating parameter on which convergence is based

=1 convergence is based on a scalar damage parameter

=2 convergence is based on effective total strain

=-10 no iterations

Enter XVIII.3. only if MDAMG = 2

XVIII.3. CMCUMAT Material Data

Line TEMP

TEMP Temperature at which the following damage properties apply

XVIII.3.a. Matrix Elastic Properties

Line  $E_m$   $\nu_m$   $f_m$   $m_m$   $Y_m$   $D_{sat}$

$E_m$	Matrix Young's Modulus [psi]
$\nu_m$	Matrix Poisson ratio
$f_m$	Matrix volume fraction ( $0 < f_m < 1$ )
$m_m$	Matrix Weibull exponent
$Y_m$	Energy release rate density [psi]
$D_{sat}$	Saturation value of matrix damage ( $0 < D_{sat} < 1$ )

XVIII.3.b. Fiber Elastic Properties

Line  $E_f$   $\nu_f$   $f_f$   $m_f$   $th_0$   $L_0$   $R$   $S_0$

$E_f$	Fiber Young's Modulus [psi]
$\nu_f$	Fiber Poisson ratio
$f_f$	Fiber volume fraction ( $0 < f_f < 1$ )
$m_f$	Fiber Weibull exponent
$th_0$	Fiber interface shear resistance [psi]
$L_0$	Fiber reference length [inch]
$R$	Fiber radius [inch]
$S_0$	Fiber reference stress [psi]

XVIII.3.c. Composite Properties

Line gamma

gamma	Orientation of composite [deg] (Enter 0., angle set internally)
-------	--

#### XVIII.3.d. Matrix Inelastic Properties for Normal Stresses

Line **m<sub>σ</sub> σ<sub>th</sub>**

a0            Slope of inelastic strain [1/psi<sup>2</sup>]

σ<sub>th</sub>            Threshold stress [psi]

#### XVIII.3.e. Matrix Inelastic Properties for Shear Stresses

Line **a1 τ<sub>th1</sub> dτ<sub>th1</sub> a2 τ<sub>th2</sub> dτ<sub>th2</sub>**

a1            First slope of inelastic strain [1/psi]

τ<sub>th1</sub>            First threshold stress [psi]

dτ<sub>th1</sub>            First stress width [psi]

a2            Second slope of inelastic strain [1/psi]

τ<sub>th2</sub>            Second threshold stress [psi]

dτ<sub>th2</sub>            Second stress width [psi]

#### XVIII.3.f. Fatigue and Creep Parameters

Line **thon N0 expn TIMEST TIMECY Ncycle**

thon

N0

expn

TIMEST

TIMECY

Ncycle    Number of cycles (input as NCYCM of Section XVIII.1.)

#### XVIII.3.g. Creep Parameters

Line **EPSID0 SIGMA0 nc**

EPSID0

SIGMA0

nc

Enter XVIII.4. only if MDAMG = 3

XVIII.4. VPI Polynomial Damage

Line **TEMP**

**TEMP** Temperature at which the following damage properties apply

Line **AA BB CC DD SA SB SC SU**

<b>AA</b>	Linear cycle term coefficient
<b>BB</b>	Quadratic cycle term coefficient
<b>CC</b>	Cubic cycle term coefficient
<b>DD</b>	Fourth order cycle term coefficient
<b>SA</b>	Stress ratio constant
<b>SB</b>	Linear stress ratio coefficient
<b>SC</b>	Quadratic stress ratio coefficient
<b>SU</b>	Normalization Stress (ultimate) [psi]

The VPI model scales elastic properties according to the following polynomial equation:

$$\frac{E}{E_0} = 1 + \left\{ A \ln(N) + B \ln(N^2) + C \ln(N^3) + D \ln(N^4) \right\} * \left\{ a + b \frac{\sigma}{SU} + c \left( \frac{\sigma}{SU} \right)^2 \right\}$$

where N is the cycle and  $\sigma$  is the stress component appropriate to the modulus being scaled.

## XIX. NODAL DISPLACEMENTS

*Enter NDIS of these lines*

Line N IDOF IFIX DISP NEND NINC

N	Node number
IDOF	Degree of freedom for which the displacement is input 1 =X translation,      2 =Y translation,      3 =Z translation, 4 =rotation about X,    5 =rotation about Y,    6 =rotation about Z
IFIX	Fixity flag 0 = free, 1 = prescribed displacement given by DISP
DISP	Prescribed displacement ( in. or rad. )
NEND	Last node having these nodal displacements
NINC	Increment on nodes from N to NEND having these nodal displacements

**NOTE:** Nodal displacements are input in the local rotated coordinate system if one has been defined for the node in the nodal skew input of Sect. III. If a local rotated coordinate system has not been defined for the node, input is in global.

## XX. NODAL APPLIED FORCES

*Enter NFOR of these lines*

Line N IDOF FORCE NEND NINC

N	Node number
IDOF	Degree of freedom for which force or moment is applied 1 =X force,              2 =Y force,              3 =Z force, 4 =moment about X,    5 =moment about Y,    6 =moment about Z
FORCE	Applied force or moment ( lbs or in-lbs, for axisymmetric elements lbs/rad or in-lbs/rad )
NEND	Last node having these nodal forces
NINC	Increment on nodes from N to NEND having these nodal forces

**NOTE:** Nodal applied forces are input in the local rotated coordinate system if one has been defined for the node in the nodal skew input of Sect. III. If a local rotated coordinate system has not been defined for the node, input is in global.



## XXI. NODAL TEMPERATURES

*Enter NTEM of the following line*

Line **N TEMP NEND NINC**

<b>N</b>	Node number
<b>TEMP</b>	Nodal temperature ( deg. F. )
<b>NEND</b>	Final node number with this temperature. Default is N
<b>NINC</b>	Node increment from N to NEND

## XXII. ELEMENT PRESSURE LOADS

*Enter NPRESS of these lines*

Line ELN FACE P1 P2 P3 P4 P5 P6 P7 P8 NEND NINC

ELN      Element number

FACE     Face number

**NOTE:** For large displacement analysis (IDIS = 1), pressures normally remain perpendicular to the element face. If FACE is entered as a negative number, the pressure will be calculated acting perpendicular to the original geometry.

P1-P8     Pressure at nodes 1-8 on face [PSI]

**NOTE:** Positive pressure causes compression in the element. If only the P1 pressure value is input applied to all nodes on the face.

NEND     Final element with this pressure. Default is ELN

NINC     Element increment from ELN to NEND

### NOTES ON FACE DEFINITIONS:

For element types 31, 32 and 33, face 1 is the N1-N2 face, face 2 is the N2-N3 face and face 3 is the N3-N1 face.

For 3D elements, faces and pressure nodes are defined according to local element connectivity as defined in the following table:

	RST AXIS	N1	N2	N3	N4	N5	N6	N7	N8
FACE 1	3	1	2	3	4	9	10	11	12
FACE 2	-3	5	6	7	8	13	14	15	16
FACE 3	2	1	2	6	5	9	18	13	17
FACE 4	-2	3	4	8	7	11	20	15	19
FACE 5	1	1	4	8	5	12	20	16	17
FACE 6	-1	2	3	7	6	10	19	14	18

### XXIII. ELEMENT ADDITION/REMOVAL OR MATERIAL CHANGE CODE

*Enter NADD of the following lines*

Line NEL MTV NEND NINC

NEL      Element number  
MTV      Material change number  
          = -1 remove element  
           $1 \leq \text{MTV} \leq \text{MMAT}$  change element to this material number  
NEND     Last element having this material change  
NINC     Increment on elements from NEL to NEND

### XXIV. FAILED STIFFNESSES

*Enter NFAIL of the following lines*

Line NEL LIST

NEL      Element with failed stiffness  
          > 0 if LIST is a list of layer numbers to have zero stiffness  
          < 0 if LIST is a list of integration points to have zero stiffness  
LIST      Layers of integration points to assign zero stiffness

**NOTE:** Shorthand notation may be used. Layer and int. pts. begin at 1 for each element with layer 1 at -LAX end of element and int.pt. 1 in layer 1 at negative ends of in plane element axes (see Section XXV for diagram). Number of int.pts. per layer = INT\*INT.

Enter XXV only if the variable ICAN of Section XI.1 is not equal to zero

#### XXV. ICAN ANALYSIS INFORMATION

##### Line NCS MHANC

NCS        > 0 Number of cross-sections to be analyzed, locations input below  
            -1 Analyze at centroid of all layered cross sections  
            -2 Analyze at all corner nodes of layered cross sections  
MHANC    Enhanced interlaminar shear stress flag for max shear stress cross sections.  
            =0 no enhanced shear stress calculations for the max shear stress cross sections  
            =1 do enhanced shear stress calculations for the max shear stress cross sections  
            =-1 do enhanced shear stress calculations and scale the enhanced shear stress profile to match the interlaminar shear stress resultant at the same cross section from FEM calculated shear stresses

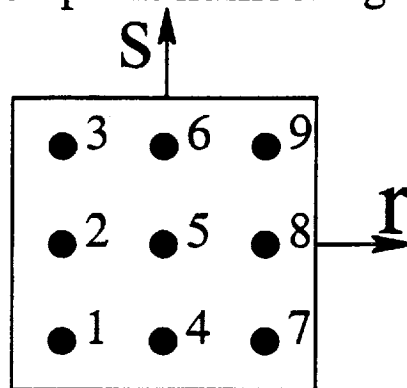
Enter NCS of the following line sets only if NCS > 0

##### Line NEL LOC NHANC

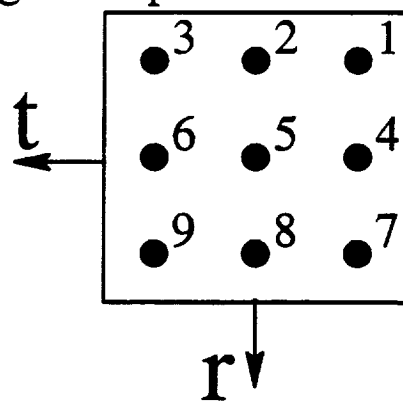
NEL        Surface element number (1st element in cross section). The cross section will be determined by locating adjacent elements moving along the LAX axis.  
LOC        Code specifying location in cross section where analysis is done.  
            =0 for centroid  
            =m for Gauss point m where m is between 1 and (integration order)\*\*2  
            Gauss points are numbered starting at negative element axes moving towards positive T, then S, then R.  
            =100 for all Gauss points  
            =-n for node number n where n is a surface node of first element in the cross section  
NHANC    Enhanced interlaminar shear stress calculation flag  
            =0 no enhanced shear stress calculations for this cross section  
            =1 do enhanced shear stress calculations for this cross section  
            =-1 do enhanced shear stress calculations and scale the enhanced shear stress profile to match the interlaminar shear stress resultant at this cross section from FEM calculated shear stresses

**NOTE:** If material axes are referenced directly to global using LTRAN of Section VII, the orientation angle is calculated assuming a global axis coincides with the through the thickness axis. This axis is found using LAX and the Jacobian (column with largest value in row LAX).

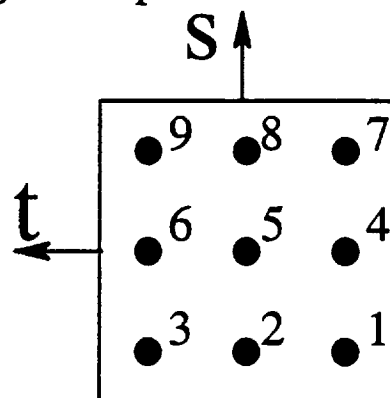
Integration point numbering for LAX = 3 (t)



Integration point numbering for LAX = 2 (s)



Integration point numbering for LAX = 1 (r)



## **5.2. HEAT TRANSFER ANALYSIS INPUT**

## HEAT TRANSFER INPUT DATA

### XXVI. HEAT TRANSFER TITLE CARD

TITLE – 80 characters max.

### XXVII. LOAD CASE CONTROL INFORMATION

**NOTE:** The heat transfer analysis type is specified in the structural input deck as variable IHT described in Section XI.1. The same applies to the method used to obtain heat transfer material properties, which is set by the variable IELA. Initial nodal temperatures are set as indicated in Section XXI. of the structural input deck.

#### XXVII.1. Analysis Options

**NHGEN NCBC NFBC NRBC NFLX ITERH HDELT NPRES NREST INT IDMH**

NHGEN 0= No internal heat generation

1= Include internal heat generation

NCBC 0= No convection heat transfer

1= Include convection heat transfer

NFBC 0= No surface heat flux

1= Include surface heat flux

NRBC 0= No radiation heat transfer

1= Include radiation heat transfer

NFLX 0= No nodal point heat flux

1= Include nodal point heat flux

ITERH Max. iterations allowed (nonlinear analyses)

HDELT Convergence criteria

The  $i$ -th iteration is converged if  $ABS \parallel (T^{(i)} - T^{(i-1)}) / T^{(i)} \parallel < HDELT$   
(Default =  $1E-6$ )

NPRES 0= No prescribed nodal temperatures

1= Read prescribed nodal temperatures

NREST 0= Not a restart

1= This is a restart case

INT Integration order.  $1 \leq INT \leq 4$

0 = default of  $(\# \text{ nodes})/8 + 1$

IDMH Heat Transfer damage flag

0 = no damage input to be read

1 = read damage inputs

Enter XXVII.2. only if *NREST* is not equal to zero

XXVII.2. Restart Option

**NTIMR RTIME**

NTIMR Time step number from which restarting

RTIME Time at the beginning of timestep NTIMR

Enter XXVII.3. for transient analysis only (*IHT* = 3 or 4)

XXVII.3. Time Step Points

**NTIMSP**

NTIMSP Number of time step points.

**TIMSP(1), TIMSP(2), ..... , TIMSP(NTIMSP)**

TIMSP(n) Starting time at each time step point.

NOTE: If NTIMSP is positive, units are [HOURS].

If NTIMSP is negative, units are [SECONDS].

**TIMINC(1), TIMINC(2), ..... , TIMINC(NTIMSP-1)**

TIMINC(n) Time increment for each time step.

(Same units as TIMSP)

NOTE: For steady state analyses (*IHT*= 1 or 2) NTIMSP is assumed to be 1.



Enter XXVIII only if not using ICAN to generate properties

### XXVIII. HEAT TRANSFER MATERIAL PROPERTIES

Enter MMAT of the following lines (MMAT = no. of different material types)

#### **IMAT NTEMP MASS**

IMAT      Material number

MASS      Mass density of material IMAT [kg/cubic cm] or [lbm/cubic inch]

NOTE: IMAT = 0 will end input if not all material properties need to be input

NOTE: MASS is initialized to the structural mass density in [lbm/cubic inch] (DEN/386.)  
input for material IMAT (See Section XIV.).

Mass densities input here will overwrite for heat transfer only.

Enter NTEMP of the following line

#### **TEMP K11 K22 K33 EMIS CP**

TEMP      Temperature [°C] or [°F]

K11,K22,K33      Thermal conductivities along material axes for the given material and temperature  
[W/m-°C] or [BTU/hr-ft-°F]

EMIS      Emissivity

CP      Specific Heat [KJ/kg-°C] or [BTU/lbm-°F]

## XXIX. HEAT TRANSFER DAMAGE

### XXIX.1. Ply Level Conductivity Damage

Line **IMAT LAW NTEM**

IMAT      Material number

LAW      Damage law to use with this material

          =0 no change in conductivity due to damage

          =1 interpolate from tables of temperature and mechanical damage

          =2 exponential damage equation

          =3 2nd order power law damage equation

NTEM     Number of temperature values at which conductivity damage will be entered

*Enter NTEM of the following two line sets, (XXIX.2. and XXIX.3.) only if LAW=1*

### XXIX.2. Conductivity Damage Table Temperatures

Line **TEM NDAM**

TEM      Temperature value

NDAM     Number of mechanical damage values  
          for which conductivity damage will be entered at this temperature

*Enter NDAM of the following line*

### XXIX.3. Conductivity Damage Tables

Line **DAM CDAM1 CDAM2 CDAM3**

DAM      Mechanical damage value

CDAM1    Conductivity damage ratio, ( $k_{dam}/k_{virgin}$ ), in material 1 direction  
          corresponding to temperature, TEM and mechanical damage, DAM

CDAM2    Conductivity damage ratio, ( $k_{dam}/k_{virgin}$ ), in material 2 direction  
          corresponding to temperature, TEM and mechanical damage, DAM

CDAM3    Conductivity damage ratio, ( $k_{dam}/k_{virgin}$ ), in material 3 direction  
          corresponding to temperature, TEM and mechanical damage, DAM

Enter NTEM of Section XXIX.4. only if LAW=2

XXIX.4. Exponential Damage Law Formulation

Line TEM

TEM      Temperature value

XXIX.4.a. Coefficient A

Line A1 A2 A3

A1      Coefficient A value for material 1 direction corresponding to temperature, TEM

A2      Coefficient A value for material 2 direction corresponding to temperature, TEM

A3      Coefficient A value for material 3 direction corresponding to temperature, TEM

XXIX.4.b. Coefficient B

Line B1 B2 B3

B1      Coefficient B value for material 1 direction corresponding to temperature, TEM

B2      Coefficient B value for material 2 direction corresponding to temperature, TEM

B3      Coefficient B value for material 3 direction corresponding to temperature, TEM

NOTE: Conductivity damage is calculated from a mechanical damage parameter, d,  
for each material direction according to the following equation:  $k_{dam} = A + (k - A)e^{-Bd}$

Enter NTEM of Section XXIX.5. only if LAW=3

XXIX.5. Power Damage Law Formulation

Line TEM

TEM Temperature value

XXIX.5.a. Coefficient A

Line A1 A2 A3

A1 Coefficient A value for material 1 direction corresponding to temperature, TEM

A2 Coefficient A value for material 2 direction corresponding to temperature, TEM

A3 Coefficient A value for material 3 direction corresponding to temperature, TEM

XXIX.5.b. Coefficient B

Line B1 B2 B3

B1 Coefficient B value for material 1 direction corresponding to temperature, TEM

B2 Coefficient B value for material 2 direction corresponding to temperature, TEM

B3 Coefficient B value for material 3 direction corresponding to temperature, TEM

NOTE: Conductivity damage is calculated from a mechanical damage parameter, d,  
for each material direction according to the following equation:  $k_{dam} = A * d * d + B * d +$

Enter XXX, only if  $NHGEN = 1$

### XXX. INTERNAL HEAT GENERATION

#### XXX.1. Element Specification

**NEL NSETQG NEND INC**

**0 (LAST LINE)**

NEL      The beginning element # in this group.  
NSETQG   Internal heat generation data set to be applied to these elements.  
NEND      Ending element in this group.  
INC      Element number increment

**NOTE:** Several of the following input sections must end with a line containing a zero (0).

This will be indicated as was done above by:

**0 (LAST LINE)**

#### XXX.2. Internal Heat Generation Data Sets

**NSETQG NFLAG NQGEND NQGINC**

**QRATE(1), ..... , QRATE(NTIMSP)**

**0 (LAST LINE)**

NSETQG   Beginning number for heat generation data sets  
NFLAG    Data set add/modify flag  
          =0 or 1 overwrite existing/create new data set  
          =2 scale (multiply) existing data set by QRATE  
          =3 add QRATE to existing data set  
NQGEND   Ending number for heat generation data sets  
NQGINC   Increment for heat generation data sets  
QRATE(j) Heat generation rate per unit volume at time point (j).  
          [Watt/(meter)<sup>3</sup>] or [BTU/(hr-ft<sup>3</sup>)].

Enter XXXI only if  $NCBC = 1$

## XXXI. CONVECTION HEAT TRANSFER

### XXXI.1. Element Specification

NEL IFACE NSETCV NEND INC

0 (LAST LINE)

NEL        The beginning element # in this group.

IFACE      The face # in this element.

(Faces are the same as for structural input. See Section XXII or XXXV.)

NSETCV    Convection data set to be applied to these elements.

NEND      The ending element in this group

INC        Element increment

### XXXI.2. Convection Data Sets

NSETCT NFLAG NCTEND NCTINC

TCV(1), ..... , TCV(NTIMSP)

HCV(1), ..... , HCV(NTIMSP)

0 (LAST LINE)

NSETCT    Beginning number for convection data sets.

NFLAG     Data set add/modify flag

=0 or 1 overwrite existing/create new data set

=2 scale (multiply) existing data set by TCV

=3 add TCV to existing data set

NCTEND    Ending number for convection data sets

NCTINC    Increment for convection data sets

TCV(j)    Temperature of the convection environment at time point (j). [ $^{\circ}\text{C}$ ] or [ $^{\circ}\text{F}$ ]

HCV(j)    Convection coefficient at time point (j). [ $\text{Watt}/(\text{meter}^2\text{-}^{\circ}\text{C})$ ] or [ $\text{BTU}/(\text{hr-ft}^2\text{-}^{\circ}\text{F})$ ]

Enter XXXII only if  $NFBC = 1$

## XXXII. SURFACE HEAT FLUX

### XXXII.1. Element Specification

**NEL IFACE NSETSF NEND INC**

**0 (LAST LINE)**

NEL      The beginning element # in this group.  
IFACE    Element face through which heat is passing  
          (Faces are the same as for structural input. See Section XXII or XXXV.)  
NSETSF   Surface heat flux data set that applies to these elements.  
NEND     The ending element in this group.  
INC      Element increment

### XXXII.2. Surface Heat Flux Data Sets

**NSETSF NFLAG NSFEND NSFINC**

**SFLUX(1), ..... , SFLUX(NTIMSP)**

**0 (LAST LINE)**

NSETSF   Beginning number for surface heat flux data sets.  
NFLAG    Data set add/modify flag  
          =0 or 1 overwrite existing/create new data set  
          =2 scale (multiply) existing data set by SFLUX  
          =3 add SFLUX to existing data set  
NSFEND   Ending number for surface heat flux data sets  
NSFINC   Increment for surface heat flux data sets  
SFLUX(j) Heat flux on the element surface at time point (j) [Watt/(meter<sup>2</sup>)] or [BTU/(hr-ft<sup>2</sup>)]

Enter XXXIII only if  $NFLX = 1$

### XXXIII. NODAL HEAT FLUX

#### XXXIII.1. Node Specifications

**NODE NSETNF NEND INC**

**0 (LAST LINE)**

NODE     The beginning node # in this group  
NSETNF   Nodal heat flux data set that applies to these nodes  
NEND     Ending node number in this group  
INC       Node number increment

#### XXXIII.2. Nodal Heat Flux Data Sets

**NSETNF NFLAG NNFEND NNFINC**

**NFLUX(1), ..... , NFLUX(NTIMSP)**

**0 (LAST LINE)**

NSETNF   Beginning number for nodal heat flux data sets  
NFLAG     Data set add/modify flag  
            =0 or 1 overwrite existing/create new data set  
            =2 scale (multiply) existing data set by NFLUX  
            =3 add NFLUX to existing data set  
NNFEND    Ending number for nodal heat flux data sets  
NNFINC    Increment for nodal heat flux data sets  
NFLUX(j)   Nodal point heat flux at time point (j) [Watts] or [BTU/hr]



Enter XXXIV only if  $NPRES = 1$

## XXXIV. PRESCRIBED NODAL TEMPERATURES

### XXXIV.1. Node Specifications

**NODE NSETPR NEND INC**

**0 (LAST LINE)**

NODE     The beginning node # in the group  
NSETPR   Prescribed temperature data set that applies to these nodes  
NEND     The ending node in this group  
INC       Node increment

### XXXIV.2. Prescribed Temperature Data Sets

**NSETPR NFLAG NPREND NPRINC**

**TFIX(1), ..... , TFIX(NTIMSP)**

**0 (LAST LINE)**

NSETPR   Beginning number for nodal prescribed temperature data sets  
NFLAG     Data set add/modify flag  
            =0 or 1 overwrite existing/create new data set  
            =2 scale (multiply) existing data set by TFIX  
            =3 add TFIX to existing data set  
NPREND   Ending number for prescribed temperature data sets  
NPRINC     Increment for prescribed temperature data sets  
TFIX(j)    The prescribed nodal temperature at time point (j) [°C] or [°F]

Enter XXXV if NRBC = 1. If NRBC = 0 go to next load case, XXVII.)

### XXXV. RADIATION HEAT TRANSFER

NOTE: Radiation heat transfer is available for IHT=2 or 4 only

#### XXXV.1. Element Specifications

**NEL IFACE VIEWFT NSETRD NEND INC**

**0 (LAST LINE)**

NEL	The beginning element # in this group
IFACE	Element face which is radiated
VIEWFT	View factor for the element face
NSETRD	The radiation data set that applies to this group
NEND	The ending element in this group
INC	Element increment

### XXXV.2. Radiation Data Sets

**NSETRT NFLAG NRTEND NRTINC NGAS**  
**TRAD(1), ..... , TRAD(NTIMSP)**  
**EMISCF GEM GAB**  
**0 (LAST LINE)**

**NSETRT** Beginning number for radiation temperature data sets

**NFLAG** Data set add/modify flag

=0 or 1 overwrite existing/create new data set

=2 scale (multiply) existing data set by TRAD

=3 add TRAD to existing data set

**NRTEND** Ending number for radiation temperature data sets

**NRTINC** Increment for radiation temperature data sets

**NGAS** Number of EMISCF GEM GAB input lines to be read for this data set (can be 0)

**TRAD(j)** The radiation environment temperature at time point (j) [°C] or [°F]

**EMISCF** Emissivity scale factor (multiplies material emissivity; =1.0 if not defined)

**GEM** Gas emissivity coefficient

=0.0 calculate using R table of Section XXXV.3.

=1.0 for surface to surface radiation (default if not defined or calculated)

**GAB** Gas absorptivity coefficient

=0.0 calculate using R table of Section XXXV.3.

=1.0 for surface to surface radiation (default if not defined or calculated)

**NOTE:** EMISCF, GEM, and GAB can be functions of temperature. The elements in the data set (Section XXXV.1.) should be of the same material. Input a line for each temperature at which the heat transfer properties of that material are defined (Section XXVIII.).

### XXXV.3. Combustion Constant Table

**TGAS(1), TGAS(2), .....**

**R(1), R(2), .....**

**TGAS(j)** Gas (flame) temperature [°F] for each R value

**R(j)** Combustion constant, used to calculate gaseous emissivity and gaseous absorptivity according to the following equations:

$$\epsilon_{gas} = 1 - \exp\left[\frac{-R}{T_{rad}^{1.5}}\right] \quad \alpha_{gas} = 1 - \exp\left[\frac{-R}{T_{wall}^{1.5}}\right]$$

**NOTE:** If gas emissivity (GEM) and gas absorptivity (GAB) are not input for a radiation data set and no combustion constant table has been entered, they take on the default value of 1.0, which corresponds to surface to surface radiation.

### NOTES ON HEAT TRANSFER INPUTS

The maximum data set number allowed is defined by the parameter MXSET as compiled in the CSTEM code. (Currently, MXSET=200.) Each data set may contain any combination of heat transfer quantities. Data set numbers need not be unique between differing quantities, i.e. it is fine to have a convection data set 1 as well as a radiation data set 1.

Element faces and face nodes are defined, in the same manner as for structural pressure loads, according to local element connectivity as defined in the following table:

	RST AXIS	N1	N2	N3	N4	N5	N6	N7	N8
FACE 1	3	1	2	3	4	9	10	11	12
FACE 2	-3	5	6	7	8	13	14	15	16
FACE 3	2	1	2	6	5	9	18	13	17
FACE 4	-2	3	4	8	7	11	20	15	19
FACE 5	1	1	4	8	5	12	20	16	17
FACE 6	-1	2	3	7	6	10	19	14	18

### **5.3. ELECTROMAGNETIC ANALYSIS INPUT AND DATA BANK**

## ABSORPTIVITY INPUT DESCRIPTIONS

### XXXVI. ELECTROMAGNETIC TITLE CARD

**TITLE:** Any 80 character title.

*Enter XXXVII. for first absorptivity load case only*

### XXXVII. MATERIAL DESCRIPTION

*Enter MMAT of the following line*

#### **MTN MATCOD**

MTN      Absorptivity material number

If MTN is negative, the data bank for this material will be printed.

**NOTE:** Absorptivity material numbers must coincide with structural geometry material numbers

MATCOD Absorptivity data bank material identifier code (1 to 8 characters long).

The identifier COND indicates a conductor material which reflects all incident radiation.

## XXXVIII. ELECTROMAGNETIC WAVE DESCRIPTION

### **NWAVE**

NWAVE Number of wave descriptions to be read

Waves are numbered consecutively according to order of input. Maximum of 10.

*Enter NWAVE of the following line*

### **NTRN WAVMAG ANGP FREQ**

NTRN Transformation number describing orientation of the wave coordinate system with respect to global

**NOTE:** Transformation info must have been input in structural section.

WAVMAG Wave magnitude (amplitude).

ANGP Polarization angle [DEG]. Measured from wave coordinate system X axis.

FREQ Wave frequency [Hz]

**NOTE:** The variable IEMAG is specified in the structural input as an Analysis Option and is a master control for absorption calculations as well as indicating the type of method to be used. The variable IABSO is specified in the structural input deck as a Load Case Control Option and is an on/off switch for the individual structures load case.

## XXXIX. PROPAGATION PATH DESCRIPTION

### **NPATH**

NPATH Number of propagation paths to be read

Paths are numbered consecutively according to order of input. ( $-10 \leq \text{NPATH} \leq 10$ )

>0 does specified paths only, does no exposure analysis

=0 determines faces exposed to incoming wave and performs absorption analysis on all faces exposed more than 90%

<0 does specified paths only, determines and prints exposed faces

*Enter ABS(NPATH) of the following line*

### **NEL FACE**

NEL Surface element which wave encounters first

FACE Element face upon which wave impinges

**NOTES:** Element faces are defined, in the same manner as for structural pressure loads, according to local element connectivity as defined in the following table:

	RST AXIS	N1	N2	N3	N4	N5	N6	N7	N8
FACE 1	3	1	2	3	4	9	10	11	12
FACE 2	-3	5	6	7	8	13	14	15	16
FACE 3	2	1	2	6	5	9	18	13	17
FACE 4	-2	3	4	8	7	11	20	15	19
FACE 5	1	1	4	8	5	12	20	16	17
FACE 6	-1	2	3	7	6	10	19	14	18

#### **XL ABSORPTIVITY LOAD CASE DESCRIPTION**

##### **NLC**

NLC      Number of load cases to be read and calculated

*Enter NLC of the following line*

##### **WAVENO PATHNO**

WAVENO Wave number to be used in this load case

PATHNO Path number to be used in this load case



## NOTES:

1) The electromagnetic wave is defined using a transformation of a "wave coordinate system" from the global coordinate system. The wave coordinate system has been defined such that the wave propagates along the wave coordinate Z (or 3) axis. Polarization is then defined as an angle relative to the wave coordinate X (or 1) axis. For example, the wave lies in the X-Z plane for 0 degree polarization, and in the Y-Z plane for 90 degree polarization.

2) The propagation path is calculated by starting at the input surface element on the impingement face then through the model by going from the entering face to the opposite face of an element and on into the adjacent element connected to that opposite face. It is assumed that the layers encountered are the same as those through the cross section at the point of impingement. The absorption is calculated using angles and temperatures calculated at the centroid of the layer incident faces within each element. Checks are made for layers extending into the next element by comparing material type and orientation of layers with the global coordinate system. If these both match the layer is skipped, thus the temperature used will have been from the surface of the segment of the layer in the first element containing that layer.

3) The element face upon which the wave impinges must be a face parallel to the layers within the element, or in other words, a face perpendicular to the through the thickness axis, LAX.

4) Two data bank formats are available to specify electric properties. The type being used is determined by counting the number of inputs on a frequency line. This means the two types can be mixed. The optional properties are converted into the standard electric property form immediately after the line is read. The equations used to make this conversion are

$$\begin{aligned}\text{PERMTYR} &= \text{DIECON} * \text{E0} \\ \text{PERMTYI} &= \text{LOSTAN} * \text{E0} \\ \text{PERMBYR} &= \text{U0} \\ \text{PERMBYI} &= 0.0 \\ \text{CONDTY} &= \text{LOSTAN} * \text{PERMTYR} * \text{FRQABS}\end{aligned}$$

with  $\text{E0}$  = free space permittivity ( $2.249\text{E}-13$  farads/in) and  $\text{U0}$  = free space permeability ( $3.192\text{E}-8$  henries/in).

## **XLI. ELECTROMAGNETIC DATA BANK FORMATS**

## XLI.1. TABLES DATA BANK FORMAT (IEMAG=3)

MATL NTEM

TEMABS(1) NFRQ

. FRQABS(1) NANG

. . ANGABS(1) NXTRA

. . . XTRABS ABSORB

. . .

. . ANGABS(NANG) NXTRA

. . XTRABS ABSORB

. FRQABS(NFRQ) NANG

. ANGABS(1) NXTRA

. . XTRABS ABSORB

. . .

. . ANGABS(NANG) NXTRA

. . XTRABS ABSORB

TEMABS(NTEM) NFRQ

FRQABS(1) NANG

. ANGABS(1) NXTRA

. . XTRABS ABSORB

. . .

. . ANGABS(NANG) NXTRA

. . XTRABS ABSORB

FRQABS(NFRQ) NANG

ANGABS(1) NXTRA

. XTRABS ABSORB

.

ANGABS(NANG) NXTRA

XTRABS ABSORB

MATL = 1 to 8 character alphanumeric material code

NTEM = Number of temperatures for this material

TEMABS = Temperature value [F]

NFRQ = Number of frequencies for this temperature

FRQABS = Frequency value [MHz]

NANG = Number of polarization angles for this frequency

ANGABS = Polarization angle [DEG]

NXTRA = 0 (this may be used for data bank expansion)

XTRABS = 0 (this may be used for data bank expansion)

ABSORB = Decimal amount of energy absorbed for this material at temperature TEMABS, frequency FRQABS, and polarization angle ANGABS.

NOTES:

- 1) The variables (NTEM, NFRQ, NANG) do not need to be the same each time they are used.
- 2) Maximums for these variables are presently: 10 for NTEM, 10 for NFRQ, 5 for NANG
- 3) MATL must begin with an alphabetic.
- 4) ABSORB is a decimal amount of energy absorbed, i.e. if all the energy is absorbed  $\text{ABSORB} = 1$   
if half the energy is absorbed  $\text{ABSORB} = 0.5$

## XLI.2. IMPEDANCE DATA BANK FORMAT (IEMAG=1 OR 2)

MATL NTEM

TEMABS(1) NFRQ

. FRQABS(1) PERMTYR PERMTYI PERMBYR PERMBYI

. .

. FRQABS(NFRQ) PERMTYR PERMTYI PERMBYR PERMBYI

TEMABS(NTEM) NFRQ

FRQABS(1) PERMTYR PERMTYI PERMBYR PERMBYI

. .

FRQABS(NFRQ) PERMTYR PERMTYI PERMBYR PERMBYI

MATL = 1 to 8 character alphanumeric material code  
NTEM = Number of temperatures for this material  
TEMABS = Temperature value [F]  
NFRQ = nNumber of frequencies for this temperature  
FRQABS = Frequency value [MHz]  
PERMTY = Complex permittivity at this temperature and frequency.  
PERMTYR real, PERMTYI imaginary  
PERMBY = Complex permeability at this temperature and frequency.  
PERMBYR real, PERMBYI imaginary

### NOTES:

- 1) The variables (NTEM and NFRQ) do not need to be the same each time they are used. In other words, the number of frequencies can be different for each temperature.
- 2) Maximums for these variables are presently: 10 for NTEM, 10 for NFRQ
- 3) MATL must begin with an alphabetic.
- 4) Conductivity (used for IEMAG=2) is calculated as:  $CONDY = PERMTYI * FRQABS$

XLI.3. DIELECTRIC CONSTANT DATA BANK FORMAT (IEMAG=1 OR 2)

MATL NTEM

TEMABS(1) NFRQ

: FRQABS(1) DIECON LOSTAN

: :

: FRQABS(NFRQ) DIECON LOSTAN

TEMABS(NTEM) NFRQ

FRQABS(1) DIECON LOSTAN

:

FRQABS(NFRQ) DIECON LOSTAN

DIECON = Dielectric constant (relative permittivity)

LOSTAN = Loss tangent

#### **5.4. ACOUSTICS ANALYSIS INPUT**

## SOUND POWER CALCULATION INPUT DESCRIPTION

### XLII. SOUND POWER CALCULATION CONTROL VARIABLES

#### NUMFRQ NDRMSK NODDIR NODFRC

NUMFRQ Number of frequencies for which radiation efficiencies and sound power are calculated

NDRMSK Number of mask points

NODDIR Direction of nodal forcing function

1 = X direction

2 = Y direction

3 = Z direction

NODFRC Node number at which force is applied

### XLIII. FREQUENCY RANGE

#### FRQMIN FRQMAX

FRQMIN Minimum frequency for sound power calculation [HZ]

FRQMAX Maximum frequency for sound power calculation [HZ]

**NOTE:** The NUMFRQ frequencies will be logarithmically distributed between FRQMIN and FRQMAX.

### XLIV. APPLIED FORCE

#### FORCE

FORCE The magnitude of the applied force [LBF]

### XLV. LOSS FACTORS

#### ETA(1) ETA(2) ... ETA(NEIG)

ETA(i) The modal loss factor for each mode. One value required for each eigenvalue calculated.

### XLVI. MASKING POINT

*Enter NDRMSK of the following line*

#### DIRMSK(1) DIRMSK(2) DIRMSK(3)

DIRMSK(i) The X, Y, and Z coordinates of the masking point.



#### NOTES:

A masking point is used to eliminate some of the surfaces of elements from being included in the sound power calculations. Faces of elements interior to a structure are automatically eliminated by the program and need not be considered for masking.

It is often desirable to not include all surfaces of a structure for sound calculations. For example the exterior surface of a pipe might be the only surface desired for sound power calculations. The interior of the pipe can be masked out by specifying a masking point anywhere on the interior of the pipe.

Masked surfaces are defined to be any element surface which has a positive dot product between the surface of an element and a vector from the element face to the masked point. Thus only one point is required to mask all interior surface faces of a straight pipe.

Output of radiation efficiencies are dimensionless and the sound power is output in [WATTS].

## **5.5. TAILORING CONTROL INPUT**

## TAILORING INPUT DESCRIPTION

### XLVII. TAILORING TITLE LINE

**TITLE:** The first line in the file is an 80 character line for identification.

### XLVIII. TAILORING CONTROL VARIABLES

**\$ NPRO** Procedure number

Designates which ANALIZ routine to use. \$ must be in column 1 followed by at least one blank space.

**NCALC NDV**

**NCALC** Specifies type of analysis

0 = read input and stop

1 = one cycle through program

2 = Optimization is performed

**NDV** Number of independent design variables.

**IPRINT ITMAX ICNDIR NSCAL ITRM**

**IPRINT** print control used in optimization.

0 = No print during optimization

1 = Print initial and final optimization information.

2 = Print above plus objective function value and design variables values at each iteration.

3 = Print above plus constraint values, direction vector and move parameter at each iteration.

4 = Print above plus gradient information.

5 = Print above plus each proposed design vector, objective function, and constraint values during the one-dimensional search.

**ITMAX** Maximum number of optimization iterations allowed

Default=20

**ICNDIR** Conjugate direction restart parameter.

Default=NDV+1.

**NSCAL** Scaling parameter. Internal scaling takes place every NSCAL times.

(0 or NDV+1 is used)

**ITRM** Number of consecutive iterations which must satisfy convergence criterion before optimization is terminated.

Default=3.

## XLIX. TOLERANCES

### **FDCH FDCHM**

**FDCH** Relative change in design variables.

Default value=.01.

**FDCHM** Minimum absolute step size.

Default = .01

### **DELFUN DABFUN ALPHAX**

**DELFUN** Minimum relative change of the objective function to indicate convergence of the optimization process.

Default = .001.

**DABFUN** Minimum absolute change for convergence.

Default = .001 times the original objective function.

**ALPHAX** Maximum fractional change in any design variable for a first estimate.

Default = .1

## L. OBJECT AND DESIGN VARIABLES

### **NDVTOT IOBJ SGNOPT**

NDVTOT Total number of variables, including variables which are linked to variables.

Default = NDV

IOBJ Global variable number associated with objective function.

This number determines the location of the objective function in GLOBAL COMMON.

SGNOPT Sign used to determine whether the objective function is to be maximized or minimized.

= +1. for maximization.

= -1. for minimization.

*Enter NDV of the next line (one for each design variable).*

### **VLB VUB X**

VLB Lower bound of design variable

If VLB .LT.  $-1.0\text{E}+15$  there is no lower bound

VUB Upper bound of design variable

If VUB .GT.  $+1.0\text{E}+15$  there is no upper bound

X Initial value for design variable.

*Enter NDV of the next line.*

### **NDSGN IDSGN AMULT**

NDSGN Design variable number associated with this input.

IDSGN Global variable number associated with this input.

AMULT Constant multiplier on this design variable

Default = 1.0

## LI. CONSTRAINTS

### **NCONS**

NCONS    Number of constraint sets.

*Enter NCONS of the next two lines*

### **ICON JCON**

ICON      First global variable number associated with constraint set.

JCON      Last global variable number associated with constraint set.

### **BL SCAL1 BU SCAL2**

BL          Lower bound for constraint

SCAL1      Normalization scale factor for BL

BU          Upper bound for constraint

SCAL2      Normalization scale factor for BU

## LII. END

### **END**

END          Indicator for end of input.

**NOTE:** The following is an example of the above input. A dollar sign (\$) in column 1 signals a comment only line, with the exception of the 2nd line. The 2nd line contains a procedure number and is used by CSTEM to branch to the subroutine associated with the procedure.

ABSORPTION TEST CASE

\$ 1 PROCEDURE 1

2,3

\$ MAX OF 3 ITERATIONS

5,3,5,5

0.5,0.5

.1

\$ OBJECT FUNCTION IS EM ABSORBPTION

3,25,1.0

\$ DESIGN VARS. ARE % OF THICKNESS FOR 3 OF 4 LAYERS

0.0,1.,.30

0.0,1.,.25

0.0,1.,.30

1,17,1.

2,18,1.

3,19,1.

\$ NO CONSTRAINTS

0

END





## **SECTION 6. DESCRIPTION OF OUTPUT**

Results from CSTEM analyses are printed out on many files. Generally, for each analysis module there is an input echo file and a printed results file, along with several scratch files. There may also be some intermediate printout files as well as some specialty files that contain data that may be used for plotting, etc. A summary of the various output files is shown [here](#). A complete listing of the CSTEM file structure is contained in Appendix 1.

### **FILE    SUMMARY**

1	STRUCTURAL AND ACOUSTICS INPUT ECHO, ERROR MESSAGES
4	STRUCTURAL RESULTS AND ELECTROMAGNETIC RESULTS
6	SYSTEM OUTPUT. CONTAINS TIMING SUMMARY, SOME ERROR MESSAGES
7	ELEMENTAL STIFFNESS FOR USE IN SUBSEQUENT RUNS
8	IN.PT. STRESSES AND STRAINS FOR USE IN SUBSEQUENT RUNS
10	STRUCTURAL RESTART FILE
18	LAYER ORIENTATION VISUALIZATION AS CENTROID DISPLACEMENTS
22	INTEGRATION POINT DATA FOR USE IN SUBSEQUENT RUNS
31	HEAT TRANSFER INPUT ECHO AND NODAL TEMP RESULTS
37	HEAT TRANSFER RESTART FILE
46	MODESHAPE STRESS/STRAIN OUTPUT FILE
51	ELECTROMAGNETIC INPUT ECHO
59	ACOUSTICS RESULTS
60	DEBUG FILE (VARIOUS INTERMEDIATE PRINT WHEN ACTIVATED)
61	ICAN RESULTS OR CROSS SECTION INTERPOLATED STRESSES
62	CROSS SECTION INTERPOLATED STRAINS
68	RAX VIBRATION RESULTS FILE
69	SIESTA TYPE VIBRATION MODESHAPES FILE
76	GENERAL TAILORING RESULTS FILE
93	TYPE SPECIFIC TAILORING RESULTS

### 6.1. INPUT ECHO, FILE1 (NT1)

This file mainly contains a labelled echo of the input read. It also contains the results of internal generation of geometry. This would include nodal coordinates and element connectivities when using the internal mesh generation and the results of element layering of representative cross sections, when applicable. Various error messages are also written to this file.

### 6.2. STRUCTURAL RESULTS, FILE4 (NT4)

If nodal banding is performed on the structural model the results are printed on this file. The max bandwidth and wavefront information for the original and banded numbering is printed with a double line under the column which was used in the analysis. If the nodes were renumbered internally, a listing of the external node numbers in the internal numbering order is written. The banding information is written only once, but all following information is written for every load case.

Data from elemental stiffness generation is printed to this file. This information is printed element by element in ascending internal order. The data which is printed can be controlled by the input variable ISTRP on the element definition line. ISTRP also controls the element stress/strain printout locations (integration point, node, face). Section V.1. of the input sheets describes the printout that can be obtained using this variable. A summation of the element volumes and weights is printed at the end of the elemental data.

Vibration frequencies and information on the convergence of large displacement iterations are printed next. Global displacements and reaction forces for all nodes listed in ascending external node number order are printed next, followed by element stress and strain results. If requested, failure criteria calculation results are printed with integration point strains. Buckling results follow the max and min stress/strain summary. The output of the displacements, forces, and stress/strain info can be controlled by the input variable IPOUT on the second load case control line as described in Section XI.1. of the input sheets. This output can be turned on and off load case by load case. In addition, the stresses and strains can be requested in global or material

axis systems. If the max and min displacements for a load case are both determined to be zero, no stress/strain output will be generated for that load case regardless of the value of IPOUT.

Electromagnetic results are also printed on this file and would appear after the structural results for a load case. If an exposure analysis was done, all element surface faces are listed along with the percentage of each surface face that can be seen by the incoming wave. The results for each wave orientation are then listed for each requested impingement location.

### 6.3. MODESHAPE STRESS/STRAIN OUTPUT, FILE46 (NT46)

This file contains stress and strain printout for each vibration modeshape calculated. Output to this file is controlled by the variable IEIGST as described in Section XI.1. of the input sheets. Modeshapes normalized to a max displacement of 1.0 are used as the structural displacements with stress recovery performed in the same manner as a static solution. The IPOUT variable determines whether a complete element by element listing is produced, or only a summary of max and min stresses and strains.

### 6.4. STRUCTURAL RESTART, FILE10 (NT10)

This file contains the necessary information to restart a run from one of the load cases on this file. This is basically control information and nodal displacements for small displacement analyses. For large displacement analyses the stress and strain information at each integration point must be saved as well. This will be a much larger file in this case. Geometry, material properties, and loading information is assumed to be contained on the input deck for a restart run. It is not contained on the restart file.

The input variable NOUT, described in Section I.3. of the input sheets, controls whether this file is created or not, and whether all load cases or only the latest load case will be saved on the file. If only the latest load case is to be saved, the restart file is rewound each time just prior to writing.

#### 6.5. ELEMENT STIFFNESS, FILE7 (NT7)

The calculation of element stiffnesses can be time consuming, so the element stiffnesses can be saved to this file and read in for a subsequent run which would have the same stiffnesses. This file is a direct access (random) file, and its use is controlled by the variable KSAVE which is read on the element definition header line of the input deck as described in Section V.1. of the input sheets.

#### 6.6. LAYER VISUALIZATION, FILE18 (NT18)

A way to visualize layer orientation is provided by printing special nodal displacements to this file. A specially numbered node is assumed to exist at each element centroid, and a displacement of this node along the fiber axis is printed for each layer of each element. The centroid node is numbered as  $2*(LX-1)+NNMAX+1$ , where LX is the element number and NNMAX is the largest node number allowed. Therefore, the nodes printed on this file do not actually exist in the model, but can be included graphically. One way to visualize the orientation is to create a zero length beam element (strut) at the element centroid and use the displacement on this file as the displacement of one of the ends of the strut. This is the reason that the nodes listed on this file are numbered by twos. The format of this file is:

Layer No., Node No., Dx, Dy, Dz, NNMAX

#### 6.7. HEAT TRANSFER INPUT ECHO AND RESULTS, FILE31 (NT31)

The input read is echoed to this file along with some of the parameters passed into the heat transfer module from the executive routine. Geometry is not printed here. Following the input echo the resulting nodal temperatures are printed in ascending external node number order for each timestep in each load case. The ending time and number of iterations required to arrive at the solution are also printed.

#### 6.8. HEAT TRANSFER RESTART, FILE37 (NT37)

This file contains basically the same nodal temperature results as on the results file, NT31, but without any labels so that it can be easily read. The results for each timestep are printed. This file is used to continue a heat transfer solution from the results at one of the stored timesteps.

#### 6.9. ELECTROMAGNETIC INPUT ECHO, FILE51 (NT51)

This file contains the echo of the electromagnetic input data. If requested by the use of a negative material number, the file also contains an echo of the data bank information for the material.

#### 6.10. ACOUSTICS RESULTS, FILE59 (NOSND)

This file contains the acoustics analysis results as radiation efficiencies and sound power generated by each calculated structural free vibration mode for each forcing frequency analyzed. The sound power for each structural vibration mode is summed and printed as a function of forcing frequency.

#### 6.11. DEBUG INFO, FILE60 (NTDBG)

Many debug statements exist in the code. Most are commented out or bypassed in some way. This file contained the printout from those statements and would contain the debug printout if these statements are reactivated.

#### 6.12. ICAN RESULTS / INTERPOLATED STRESS, FILE61 (NT61)

This is the output file from the ICAN program when an ICAN micromechanics analysis is requested as described in Section XI.1. of the input sheets (variable ICAN = 1, 2, -1, or -10). It is the same as described in the ICAN User Manual, which is included as an appendix to this manual, with the exception that the location on the model for which the analysis was performed

is listed at the beginning of each results summary. The location at which a microanalysis is to be done is specified as described in Section XXV. of the input sheets.

If a cross section interpolation only is requested (variable ICAN = 3 or 4), this file contains the interpolated stresses in each ply at the cross section location specified in Section XXV. of the input sheets. The stresses are obtained by interpolating strains from the integration points of each element contained in the cross section to the upper, middle, and lower surfaces of each ply at the cross section location, then calculating the stress using the material constitutive matrix for the ply material. Interlaminar shear stresses (SIG23 and SIG31) are calculated from equilibrium equations.

#### 6.13. INTERPOLATED CROSS SECTION STRAIN, FILE62 (NT62)

This file is printed if a cross section interpolation is requested by the input variable ICAN on the load case control line, as described in Section XI.1. of the input sheets. This file contains strains at each layer upper, middle, and lower surfaces through the cross section. These values are interpolated from the integration point strain results of each ply within an element using a Lagrange polynomial approach. The strains can be requested in either the global or material axis system.

#### 6.14. RAX VIBRATION RESULTS, FILE68 (NFPD)

This file is used in the modeshape slope calculations. It contains nodal masses and resulting natural frequencies and mode shapes. Only the last load case with vibration analysis is contained on this file as it is rewound before writing.

#### 6.15. SIESTA VIBRATION MODE SHAPES, FILE69 (NT69)

This file contains the natural frequencies and normalized mode shapes in the form for use with the postprocessing portion of the SIESTA computer program. This form is fairly generic and can easily be adapted for use in many postprocessing programs. The results for all modes for every load case that performed a vibration analysis will be contained on this file.

## 6.16. GENERAL TAILORING RESULTS, FILE76 (I6I)

This file contains the generic results of a tailoring analysis. The variables are not identified as to what they actually represent in the physical structure; it is the direct output from the COPES / CONMIN program itself.

## 6.17. SPECIFIC TAILORING RESULTS, FILE93 (93)

This file contains the tailoring results with the variables identified. This is an analysis specific printout. The specific subroutines which are called to fill the tailoring variables from the finite element solution write to this file.

## 6.18. SCRATCH FILES

Many scratch files are used in CSTEM, most of which are not of real interest to the user. They contain information and results of intermediate calculations that are necessary to perform the requested type of analysis. A complete listing of the CSTEM file structure is contained in Appendix 1. There are a few of these scratch files that can be reused in certain cases to reduce the amount of computation necessary to perform an analysis.

### 6.18.1. Element Stiffness and Integration Point Data, FILE7, FILE22, FILE42

The files, NT7 and NT22, (and sometimes NT42 for very large jobs) can be used together to eliminate calculation of element stiffnesses and integration point quantities (such as the material matrix and the strain displacement matrix) in cases when the stiffness either does not change at all or, as in the case of a large displacement analysis, the stiffness is initially the same as in a previously performed analysis. Things that would change the stiffness such that this could not be done include changes in geometry, temperature, material properties, and integration order. The use of these files is indicated through the variable KSAVE, described in Section V.1. of the input sheets. The manner in which these files are saved, and also how they are input to a subsequent run, is through the use of JCL. After execution of the program, these files should be made permanent so that they can be used later. Note that on a CRAY, these scratch files may

have been written to SSD or some other media that may require copying the files to disk, then saving them. On a subsequent run, they should be made local to the job with file names FILE7 and FILE22 (and FILE42 if the spill file was used).

#### 6.18.2. Integration Point Constitutive Data, FILE8

File NT8 can be used by the micromechanics portion of CSTEM to do further processing after a previous structural finite element analysis that generated the file. This can be done by indicating the number of load cases (NLC) as a negative value and attaching the previously saved integration point constitutive file as FILE8 using the JCL controlling the run. The types of further analysis performed might include interpolation of stresses and strains to various different cross sections in either global or material systems.

#### 6.19. PATRAN RESULTS FILES

ASCII PATRAN results files can be output if so indicated by the variable IPAT. These files can be used to display results in PATRAN, given that a PATRAN description of the model geometry exists. Multiple load case results are handled by generating multiple files. The existence of any files with the generic name is checked. If any exist, a number is appended to the generic filename until a unique filename.appendix combination is found. Thus, a file with the generic name is the first file (presumably the first load case) with subsequent files created as NAME.2, NAME.3, etc.

##### 6.19.1. Displacement Results File, PATRANDISP

This file contains nodal displacement information in the form of node number and global displacements for each degree of freedom for the node. Only the three translation degrees of freedom are printed.

##### 6.19.2. Nodal Temperature Results File, PATRANTEMP

This nodal results file contains nodal temperature information in the form of node number and temperature. Temperatures are input temperatures for structural only analyses or calcu-



lated nodal temperatures for heat transfer analyses. A temperature file is created for each structural load case or at the end of each heat transfer load case.

### 6.19.3. Nodal Results File, PATNDRES

This nodal results file contains averaged nodal stresses in the form of node number, effective stress, and six stress components in the order  $\sigma_{xx}$ ,  $\sigma_{yy}$ ,  $\sigma_{zz}$ ,  $\tau_{xy}$ ,  $\tau_{yz}$ , and  $\tau_{zx}$ . Therefore, column 1 is effective stress, column 2 is  $\sigma_{xx}$  (or  $\sigma_{11}$ ), column 3 is  $\sigma_{yy}$ , etc. to column 7 is  $\tau_{zx}$ . Nodal strains are on the same file in columns 8 through 14 in the order effective strain followed by the components  $\epsilon_{xx}$ ,  $\epsilon_{yy}$ ,  $\epsilon_{zz}$ ,  $\epsilon_{xy}$ ,  $\epsilon_{yz}$ , and  $\epsilon_{zx}$ .

A nodal stress and strain results file is generated for homogeneous (unlayered) models only. The reference frame for nodal stresses and strains is as indicated by the variable IPOUT of Section XI.1. Nodal averages are unweighted averages. Stress and strain results at element integration points are extrapolated to the nodes connected to the element using Lagrange interpolating polynomials. The accumulated stresses and strains for each node are then averaged by the number of elements to which each node is connected.

### 6.19.4. Element Stress Results File, PATELSIG

This element results file contains stresses calculated at element centroids for unlayered elements or at the centroid of each layer within an element for layered elements. The reference frame for element stresses is indicated by the variable IPOUT as described in Section XI.1. of the input sheets. The stresses contained in this file can be used in conjunction with the transformations contained on the Element Coordinate System file (PATRANCRD) to display element vector/tensor plots. Note that in PATRAN, it is the responsibility of the user to match the layer transformation on the Element Coordinate System file to the appropriate columns of the Element Stress Results file. The Element Stress Results file for layered elements contains all six stress components for each layer in sequence starting with layer 1, which is at the negative end of the element stacking axis. Thus, stresses in layer 1 of an element are in columns 1–6 in the order  $\sigma_{11}$ ,  $\sigma_{22}$ ,  $\sigma_{33}$ ,  $\tau_{12}$ ,  $\tau_{23}$ , and  $\tau_{31}$ , with layer 2 in columns 7–12, layer 3 in columns 13–18, etc.

#### 6.19.5. Element Coordinate System File, PATRANCRD

This file contains transformations orienting the material coordinate system of each layer within an element to the global coordinate system. This file is used in conjunction with the Element Stress Results file (PATELSIG) to display element vector/tensor plots. The transformations contained on this file are numbered layer by layer, with layer 1 beginning at the negative end of the element stacking axis.

#### 6.19.6. Modeshape Displacement Files, PATFREQ and PATBUCK

Modeshapes are contained in PATRAN displacement results files, one file for each mode. The vibration modeshapes are written to files PATFREQn where n is a mode number. The buckling modeshapes are written to files PATBUCKn where n is the buckling mode number.

#### 6.19.7. Damage Parameter File, PATDAMGP

Damage parameters for the CMCUMAT damage mechanics model are contained in a PATRAN element results file called PATDAMGP. The Damage Parameter file for layered elements contains all damage parameters for each layer in sequence starting with layer 1, which is at the negative end of the element stacking axis. The damage parameters consist of 4 matrix damage parameters and 2 fiber damage parameters. Since the CMCUMAT model pertains to woven materials, damage is quantified in 2 directions and 2 layers. The Dm00 parameters are matrix damage parameters for a unidirectional layer oriented in the material 1 direction and the Df00 parameter is a fiber damage parameter for a unidirectional layer oriented in the material 1 direction. The Dm90 parameters are matrix damage parameters for a unidirectional layer oriented in the material 2 direction and the Df90 parameter is a fiber damage parameter for a unidirectional layer oriented in the material 2 direction. The matrix damage parameters for each layer have components in both the material 1 (0) and material 2 (90) direction. The order of the parameters on the PATDAMGP file are Dm00(1), Dm00(2), Df00, Dm90(1), Dm90(2), Df90.

**APPENDIX 1.**  
**CSTEM FILE STRUCTURE**



## CSTEM FILE DESCRIPTION

<u>NAME</u>	<u>DESCRIPTION</u>	<u>STATUS</u>	<u>TYPE</u>	<u>ACCESS</u>	<u>OPEN</u>
NT1	INPUT ECHO WITH HEADINGS	PO	SA	-	INIT1
NT2	USER'S INPUT DECK	IP	SA	-	INIT1
NT3	UNMODIFIED GLOBAL STIFF	SC	SB	-	OPSTIF
NT4	PRINTED OUTPUT	PO	SA	-	INIT1
NTIN	SYSTEM INPUT	IP	SA	-	
NTOUT	SYSTEM OUTPUT	PO	SA	-	
NT7	ELEMENTAL STIFFNESS	SC	RA	STIFIO	OPELEM
NT8	INT PT CONSTITUTIVE DATA	SC	RA	CMODIO	
NT9	NODE DATA	SC	RA	NODEIO	OPGEOM
	ELEMENT DATA			ELEMIO	
	TRANSFORMATION MATRICES			TRANIO	
NT10	OUTPUT RESTART FILE	RR	SA		OPFILE
NT11	CONSTRAINT EQN STORAGE	SC	RA	CSTIO	OPGEOM
NT12	INPUT RESTART FILE	IP	SA	-	INIT1
NT13	ASSEM'D NONLIN. STIFFNESS	SC	SB	-	OPSTIF
NT14	MODIFIED ORIG. STIFFNESS	SC	SB	-	OPSTIF
NT16	RESTART USER DATA FILE48	IP	SA	REUSER	OPREDU
NT17	SAVED ELEMENTAL STIFFNESS	RR	RA	-	OPELEM
NT18	LAYER ORIENTATIONS	PO	SA	-	OPELEM
NT19	MATERIAL LAYER INFO	SC	RA	LAYIO	OPGEOM
	ELEMENT PRESSURE DATA	SC	RA	PRESIO	
NT20	PREGENERATED GEOMETRY	IP/C	SA	-	RDGEOM
NT20	ELEMENTAL FORCES (6)	SC	RA	ELFOIO	OPELEM
	ELEMENT EQUATION NUMBERS	SC	RA	ELEQIO	
	ELEMENT LUMPED MASS	SC	RA	LMSIO	
NT21	PREGENERATED TRANSFORMS	IP/C	SA	-	RDGEOM
NT22	PREGENERATED LAYER INFO	IP/C	SA	-	RDGEOM
NT22	INT POINT STRESS RECOVERY	SC	RA	STREIO	OPELEM
NT23	FORCES (5)	SC	RA	FORCIO	OPDISP
	DISPLACEMENTS (2)	SC	RA	DISPIO	
	LUMPED MASS	SC	RA	MASSIO	

<u>NAME</u>	<u>DESCRIPTION</u>	<u>STATUS</u>	<u>TYPE</u>	<u>ACCESS</u>	<u>OPEN</u>
NT24	MATERIAL DATA	SC	RA	MDATIO	OPFILE
NTHYD (26)	INHYD DATA FILE	SC	SA	-	RDCAN
NTBANK (27)	ICAN DATA BANK	IP	SA	BANKRD	RDCAN
NSTF (28)	MODIFIED GLOBAL STIFF	SC	SB	-	OPSTIF
NRED (29)	DECOMPOSED STIFFNESS	SC	RA	-	OPSTIF
NT30	HEAT TRANSFER TEMP. DATA	SC	RA	TEMPX	OPHEAT
NT31	HEAT TRANSFER OUTPUT	PO	SA	-	OPEMH (HINP1)
NT32	HEAT TRANSFER INPUT DECK	IP	SA	-	OPEMH (HINP1)
NT33	HEAT TRANSFER RESTART	IP	SA		JCL
NT34	H.T. ELEMENT FACE B.C.s	SC	RA	ELEMBC	OPHEAT
NT35	H.T. B.C. DATA SETS	SC	RA	BCDATA	OPHEAT
NT37	H.T. OUTPUT RESTART FILE	PO	SA		JCL
NT42	SPILL FILE ( FOR INT PT FILE NT22 )	SC	RA	STREIO	OPELEM
NT43	LAYER GENERATION	SC/C	SA	-	RADII
NT44	MESH GENERATOR INPUT (TIMESHARE ONLY)	IP	SA	-	RADMAN
NT45	X-SECTION LAYUP STORAGE	SC	SB	-	LAYUP
NT46	MODESHAPE STRESS/STRAIN	PO	SA	SIGMOD	OPMODE
NT47	USER MATERIAL DATA FILE	SC	RA	UMATIO	OPUMAT
NT48	USER CONSTITUTIVE DATA FILE	SC	RA	HOOKIO	OPUMAT
NT49	AVG NODAL STRESS/STRAIN	SC	RA	ADDSIG/AVGN	OPAVG
NTABS (50)	EM ABSORPTIVITY DATA BANK	IP	SA	-	ABINP1
NT51	ABSORPTIVITY OUTPUT	PO	SA	-	OPEMH (ABINP1)
NT52	ABSORPTIVITY INPUT DECK	IP	SA	-	OPEMH (ABINP1)
NT53	SURFACE FACES AND NODES	SC	SA	-	OPEMH (FNDSUR)

<u>NAME</u>	<u>DESCRIPTION</u>	<u>STATUS</u>	<u>TYPE</u>	<u>ACCESS</u>	<u>OPEN</u>
INSND (54)	ACOUSTIC INPUT DECK	IP	SA	SNDINP	OPSND
ISCRA (55)	ACOUSTIC WORK FILE A	SC	SB	-	OPSND
ISCRB (56)	ACOUSTIC WORK FILE B	SC	SB	-	OPSND
NTMODE (57)	MODAL DISPL AND FREQ	SC	RA	MODEIO	OPMODE
NTSRF (58)	SURFACE ELEMENTS	SC	SB	RECOM	OPSND
NOSND (59)	ACOUSTIC PRINTED OUTPUT	PO	SA	SNDPWR	OPSND
NTDBG (60)	DEBUG	PO	SA	-	INIT
NT61	ICAN OUTPUT	PO	SA	-	OPEMH (CANMAN)
NT62	X-SEC INTERP STRESS-STRAIN	PO	SA		OPEMH (CANMAN)
NT63	CONSISTENT MASS WORK FILE	SC	SB	COMPSC	OPEIG
NTE1 (64)	DET. SEARCH WORK FILE (EIGENVECTORS)	SC	SB	-	OPEIG
NTE2 (65)	SUBSPACE ITER. WORK FILE (STIFFNESS)	SC	SB	-	OPEIG
IOUT (66)	EIG. ANALYSIS DETAIL PRINT	PO	SA	-	OPEIG
NMSS (67)	CONSISTENT MASS FILE	SC	SB	-	OPSTIF
NT68	RAX FILE	PO	SA	WRMOD	OPEIG
NT69	SIESTA MODE SHAPE FILE	PO	SA	WRMOD	OPEIG
PLY.UOF (69)	SIESTA PLY UOF FILE	PO	SA		CANMAN

<u>NAME</u>	<u>DESCRIPTION</u>	<u>STATUS</u>	<u>TYPE</u>	<u>ACCESS</u>	<u>OPEN</u>
I5I (75)	TAILORING INPUT FILE	IP	SA		JCL
I6I (76)	GENERAL TAILORING OUTPUT	PO	SA		JCL
ISCR1 (77)	TAILORING SCRATCH FILE	SC	SA		JCL
ISCR2 (78)	TAILORING SCRATCH FILE	SC	SB		JCL
SLOP (91)	MODESHAPE SLOPE INPUT	IP	SA		WRMOD
93	SPECIFIC TAILORING OUTPUT	PO	SA		JCL
95-99	PATRAN OUTPUT FILES	PO	SA		OPAT

TYPE:

SA = SEQUENTIAL ASCII

SB = SEQ BINARY

RA = RANDOM ACCESS

STATUS:

IP = INPUT

PO = PRINTED OUTPUT

RR = KEEP FOR RESTART

SC = SCRATCH

/C = CLOSED AFTER USE



**APPENDIX 2.**  
**WAVES USER MANUAL**



**A COMPUTER PROGRAM FOR DETERMINING THE REFLECTION  
AND TRANSMISSION PROPERTIES OF MULTIPLAYER  
PLANE IMPEDANCE BOUNDARIES**

**CHARLES H. KRUEGER, JR.**

## ABSTRACT

This report discusses a computer program which was developed to compute the reflection and transmission coefficients of multilayer plane boundaries. The layers may have arbitrary dielectric and magnetic properties; the only restriction is that these properties be homogeneous and isotropic. The reflection and transmission coefficients are computed for arbitrary incidence angle and for the boundary terminated in both free space and in a short circuit. The coefficients are computed for two orthogonal incident signal polarizations (electric field perpendicular and parallel to the plane of incidence).

This abstract is subject to special export controls and each transmittal to foreign governments or foreign nationals may be made only with prior approval of the Air Force Avionics Laboratory (AVWE), Wright-Patterson Air Force Base, Ohio.

## INTRODUCTION

This report discusses a computer program which was developed for studying the electromagnetic wave transmission and reflection properties of radome type materials. The developed program permits prediction of the complex transmission and reflection coefficients of a uniform plane wave from a plane boundary. The boundary may consist of a finite number of layers of arbitrary thickness and arbitrary dielectric and magnetic properties. The only restriction is that the dielectric and magnetic properties must be homogeneous and isotropic within individual layers. Features such as arbitrary angle of incidence and frequency are also provided. For each boundary configuration and incident signal configuration, the following three calculations are performed for two orthogonal signal polarizations (electric vector perpendicular and parallel to the plane of incidence); transmission and reflection coefficients with boundary terminated in free space and reflection coefficients with boundary terminated in a short circuit.

The program is well suited for studying a wide variety of boundary scattering problems. The program may be used for computing the reflection and transmission of multilayer structures in coaxial transmission lines or wave guides by a suitable interpretation of the angle of incidence.

Note that this copy of this report has been modified slightly for use in the CSTEM User's Manual. The original report is Technical Report AFAL-TR-67-191, and was obtained from the Defense Technical Information Center.

## THEORY

A wave matrix formulation was selected for this program because the approach is straightforward and yields a method of solution which can be modified easily to treat many types of problems. An excellent discussion of this formulation may be found in the book Field Theory of Guided Waves, by Robert E. Collin, published in 1960 by McGraw-Hill Book Co.

Consider an electromagnetic wave incident at an angle  $\theta_i$  on the plane boundary between two media of different dielectric and magnetic properties as shown in Figure X1.1 . The incident wave is polarized with the electric field intensity oriented perpendicular to the plane of incidence. The plane of incidence is defined by the normal  $n$  to the boundary and the direction of propagation  $Z$  of the incident wave. The medium on the left of the boundary is free space, and the medium on the right is a homogeneous isotropic material whose relative permittivity is  $K_e$ , and the relative permeability is  $K_m$  where

$$\begin{aligned} K_e &= \epsilon/\epsilon_0 \\ K_m &= \mu/\mu_0 \\ \mu &= \text{complex permeability of medium} \\ \mu_0 &= \text{permeability of free space} \\ \epsilon &= \text{complex permittivity of medium} \\ \epsilon_0 &= \text{permittivity of free space} \end{aligned}$$

The incident field is given by

$$E_x = e^{-jk_{\text{sub}}0z} \quad ; \quad H_y = \frac{k_0}{\mu_0\omega} e^{-jk_0z} \quad (1.)$$

where

$$\begin{aligned} k_0 &= \omega \sqrt{\mu_0\epsilon_0} = \frac{2\pi}{\lambda_0} = \text{free space wave number} \\ \lambda_0 &= \text{wavelength of incident wave in free space} \end{aligned}$$

Now expressing the incident wave components in the  $v, u, x$  coordinates, we have

$$\begin{aligned} y &= v \cos \theta_i + u \sin \theta_i \\ z &= -v \sin \theta_i + u \cos \theta_i \\ x &= x \end{aligned}$$

so that

$$E_x = \exp[-jk_0(-v \sin \theta_i + u \cos \theta_i)] \quad (2.)$$

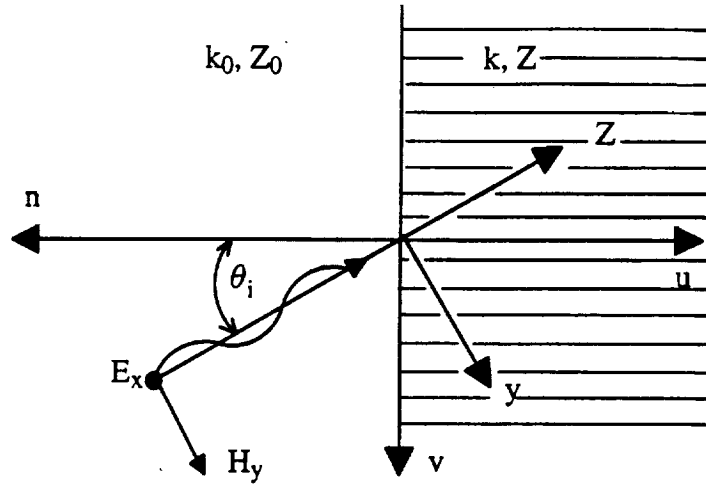


FIGURE X1.1 Boundary Between Free Space and Medium With Relative Permittivity  $K_e$  and Relative Permeability  $K_m$

To find  $\vec{H}$  we take the curl of  $\vec{E}$  in the new coordinates

$$\nabla \times \vec{E} = -j\omega\mu_0\vec{H}$$

$$j\omega\mu_0 H_u = jk_0 \sin \theta_i E_x \quad (3.)$$

$$j\omega\mu_0 H_y = jk_0 \cos \theta_i E_x$$

$$Z_a = \frac{E_x}{H_y} = \sqrt{\frac{\mu_0}{\epsilon_0}} \sec \theta_i \quad (4.)$$

where  $Z_a$  is the wave impedance of free space in a direction normal to the boundary. When the wave passes through the boundary surface, it is bent or refracted at an angle  $\theta_r$  to the boundary normal. With respect to the  $v, u, x$  coordinate system, the form of the wave will be the same as in air (Equation 4.) except for replacing  $\epsilon_0$  by  $K_e\epsilon_0$ ,  $\mu_0$  by  $K_m\mu_0$ ,  $\theta_i$  by  $\theta_r$ , and  $k_0$  by  $k$ , where

$$k = \sqrt{K_e K_m} k_0 = \frac{2\pi}{\lambda_0} \sqrt{K_e K_m} \quad (5.)$$

= wave number in material

In order that the tangential components of the fields match on the boundary, it is necessary that the variation of the field in the  $V$  direction be the same on both sides of the

boundary. This requires that

$$k_0 \sin \theta_i = k_0 \sqrt{K_e K_m} \sin \theta_r \quad (6.)$$

The wave impedance on the right side of the boundary may now be written for perpendicular polarization by analogy with the preceding free space expression, Equation 4.

$$Z_b = \sqrt{\frac{\mu_0 K_m}{\epsilon_0 K_e}} \sec \theta_r \quad (7.)$$

If we now normalize  $Z_b$  with respect to the free space impedance  $Z_a$ , we have

$$Z = \frac{Z_b}{Z_a} = \frac{\sqrt{\frac{K_m \mu_0}{K_e \epsilon_0}} \sec \theta_r}{\sqrt{\frac{\mu_0}{\epsilon_0}} \sec \theta_i} = \sqrt{\frac{K_m}{K_e}} \frac{\sec \theta_r}{\sec \theta_i} \quad (8.)$$

From the tangential boundary conditions, we have from Equation 6.

$$\begin{aligned} \sin \theta_r &= \frac{\sin \theta_i}{\sqrt{K_e K_m}} \\ \sec \theta_r &= \frac{1}{\sqrt{1 - \sin^2 \theta_r}} = \frac{1}{\sqrt{1 - \frac{\sin^2 \theta_i}{K_m K_e}}} \\ \sec \theta_r &= \frac{\sqrt{K_e K_m}}{\sqrt{K_m K_e - \sin^2 \theta_i}} \end{aligned} \quad (9.)$$

Therefore,

$$Z = \sqrt{\frac{K_m}{K_e}} \frac{\sqrt{K_e K_m} \cos \theta_i}{\sqrt{K_m K_e - \sin^2 \theta_i}} = \frac{K_m \cos \theta_i}{\sqrt{K_e K_m - \sin^2 \theta_i}} \quad (10.)$$



The propagation factor in and normal to the boundary is given by

$$\phi = k_0 \sqrt{K_e K_m} \cos \theta_r = \frac{2\pi}{\lambda_0} \sqrt{K_e K_m - \sin^2 \theta_i} \quad (11.)$$

We may deduce the impedance and propagation expressions appropriate to parallel polarization by application of Babinet's principle to the preceding results. This is accomplished by the transformation (primed quantities will refer to parallel polarization)

$$\begin{aligned} \bar{E}' &= \sqrt{\frac{\mu}{\epsilon}} \bar{H} \\ \bar{H}' &= -\sqrt{\frac{\epsilon}{\mu}} \bar{E} \end{aligned} \quad (12.)$$

so that on the right side of the boundary

$$\begin{aligned} \bar{E}' &= \sqrt{\frac{K_m \mu_0}{K_e \epsilon_0}} \bar{H} \\ \bar{H}' &= -\sqrt{\frac{K_e \epsilon_0}{K_m \mu_0}} \bar{E} \\ \frac{\bar{E}'}{\bar{H}'} &= -\sqrt{\frac{K_m \mu_0}{K_e \epsilon_0}} \frac{\bar{H}}{\bar{E}} \end{aligned}$$

Then

$$\begin{aligned} Z_a' &= -\sqrt{\frac{\mu_0}{\epsilon_0}} \cos \theta_i \\ Z_b' &= -\sqrt{\frac{k_m \mu_0}{K_e \epsilon_0}} \cos \theta_r \end{aligned}$$

so that

$$Z' = \frac{2\pi}{\lambda_0} \sqrt{K_e K_m - \sin^2 \theta_i} \quad (13.)$$

The propagation factor for parallel polarization is the same as that for perpendicular polarization. Therefore,

$$\phi' = \frac{2\pi}{\lambda_0} \sqrt{K_e K_m - \sin^2 \theta_i}$$

The next step in this development is to consider a chain matrix formulation for the reflection and transmission of waves from plane boundaries. We will define the coefficients in the chain formulation in terms of the propagation factor and characteristic or wave impedances developed above.

According to Collin, consider the plane discontinuity interface shown in Figure X1.2. We have two TEM (transverse electric mode) waves  $c_1$  and  $b_2$  approaching the boundary from left and right, respectively. A portion of  $c_1$  passes through the interface ( $T_{12}c_1$ ) into medium 2 and a portion of  $c_1$  is reflected back into medium 1 ( $R_1c_1$ ). Similarly on the other side of the boundary, a portion of  $b_2$  passes through the boundary ( $T_{21}b_2$ ) and a portion is reflected ( $R_2b_2$ ). Let  $b_1$  represent the sum of the wave components traveling away from the boundary in medium 1 and let  $c_2$  represent the sum of the wave components traveling away from the boundary in medium 2. Then

$$\begin{aligned} b_1 &= T_{21}b_2 + R_1c_1 \\ c_2 &= T_{12}c_1 + R_2b_2 \end{aligned} \quad (14.)$$

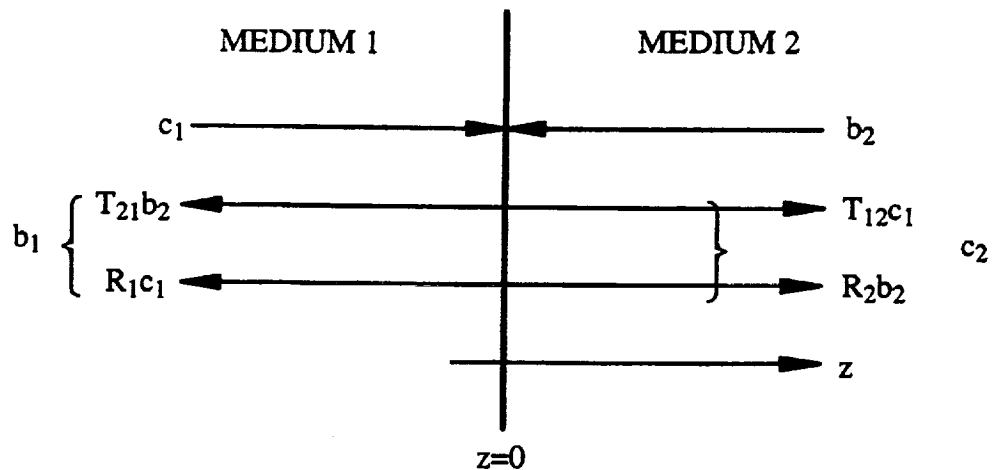


FIGURE X1.2 Plane Discontinuity Interface Showing Notation for Chain Matrix Formulation

Rearranging we find

$$\begin{aligned} b_1 &= \left( \frac{T_{12}T_{21} - R_1R_2}{T_{12}} \right) b_2 + \left( \frac{R_1}{T_{12}} \right) c_2 \\ c_1 &= \left( \frac{1}{T_{12}} \right) c_2 - \left( \frac{R_2}{T_{12}} \right) b_2 \end{aligned} \quad (15.)$$

Expressing these two equations in matrix form, we have

$$\begin{aligned} \begin{bmatrix} c_1 \\ b_1 \end{bmatrix} &= \frac{1}{T_{12}} \begin{bmatrix} 1 & -R_2 \\ R_1 & T_{12}T_{21} - R_1R_2 \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \\ &= \begin{bmatrix} A_{11} & A_{12} \\ A_{21} & A_{22} \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \end{aligned} \quad (16.)$$

where

$$\begin{aligned} A_{11} &= \left. \frac{c_1}{c_2} \right|_{b_2=0} = \frac{1}{T_{12}} \\ A_{12} &= \left. \frac{c_1}{b_2} \right|_{c_2=0} = \frac{-R_2}{T_{12}} \\ A_{21} &= \left. \frac{b_1}{c_2} \right|_{b_2=0} = \frac{R_1}{T_{12}} \\ A_{22} &= \left. \frac{b_1}{b_2} \right|_{c_2=0} = \frac{T_{12}T_{21} - R_1R_2}{T_{12}} \end{aligned} \quad (17.)$$

If we now define  $R_1$ ,  $R_2$ ,  $T_{12}$ ,  $T_{21}$  in terms of the characteristic impedances of medium 1 and medium 2 ( $Z_1$  and  $Z_2$ , respectively) we have

$$\begin{aligned} R_1 &= \frac{Z_2 - Z_1}{Z_2 + Z_1} \quad ; \quad T_{12} = 1 + R_1 \\ R_2 &= \frac{Z_1 - Z_2}{Z_1 + Z_2} \quad ; \quad T_{21} = 1 + R_2 \end{aligned} \quad (18.)$$

where  $Z_1$  and  $Z_2$  are defined in either Equation 10. or 13. for perpendicular and parallel polarization, respectively. Then

$$\begin{aligned} R_1 &= -R_2 \\ T_{12}T_{21} - R_1R_2 &= 1 \end{aligned}$$

and

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \frac{1}{T_{12}} \begin{bmatrix} 1 & R_1 \\ R_1 & 1 \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \quad (19.)$$

To apply this notation to a multilayered boundary, we need to find a relationship which describes the manner in which the wave is attenuated and phase-shifted as it passes through an individual layer. Such a relationship is provided by the expressions

$$\begin{aligned} c_1 &= \exp(j\phi d) c_2 \\ b_1 &= \exp(-j\phi d) b_2 \end{aligned} \quad (20.)$$

or expressed in matrix form

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \begin{bmatrix} e^{j\phi d} & 0 \\ 0 & e^{-j\phi d} \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \quad (21.)$$

where  $d$  is the distance between the planes where  $c_1$ ,  $b_1$ , and  $c_2$ ,  $b_2$  are evaluated. The quantity  $\phi$  is the complex propagation factor developed earlier for arbitrary angle of incidence (Equation 11.).

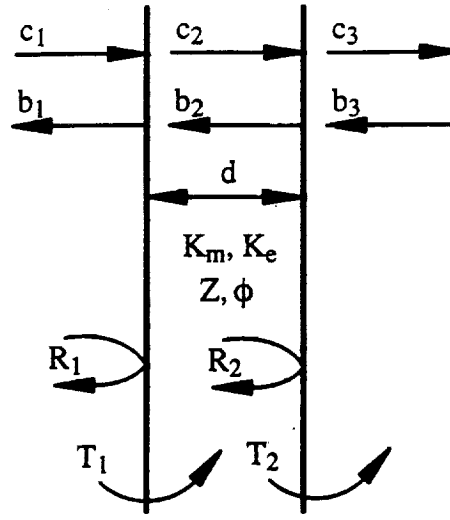


FIGURE X1.3 Two Layer Impedance Boundary

We may now use this notation to describe the reflection and transmission properties of a two layer plane impedance boundary as shown in Figure X1.3 . The first layer has a thickness  $d$  and is made of a material with complex permittivity  $K_e$  and complex permeability  $K_m$ . The characteristic impedance and propagation factor of the first layer are  $Z$  and  $\phi$  , respectively. From the preceding, we may express the relationship between  $c_1$ ,  $b_1$  and  $c_2$ ,  $b_2$  as

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \frac{1}{T_1} \begin{bmatrix} 1 & R_1 \\ R_1 & 1 \end{bmatrix} \begin{bmatrix} e^{j\phi d} & 0 \\ 0 & e^{-j\phi d} \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \quad (22.)$$

or

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \frac{1}{T_1} \begin{bmatrix} e^{j\phi d} & R_1 e^{-j\phi d} \\ R_1 e^{j\phi d} & e^{-j\phi d} \end{bmatrix} \begin{bmatrix} c_2 \\ b_2 \end{bmatrix} \quad (23.)$$

If we have a three layer boundary, then  $c_2$ ,  $b_2$  would similarly be related to  $c_3$ ,  $b_3$  so that we have (Figure X1.4 ) for an  $n$  layer boundary

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \prod_{i=1}^n \frac{1}{T_i} \begin{bmatrix} e^{j\phi_i d_i} & R_i e^{-j\phi_i d_i} \\ R_i e^{j\phi_i d_i} & e^{-j\phi_i d_i} \end{bmatrix} \begin{bmatrix} c_{n+1} \\ b_{n+1} \end{bmatrix} \quad (24.)$$

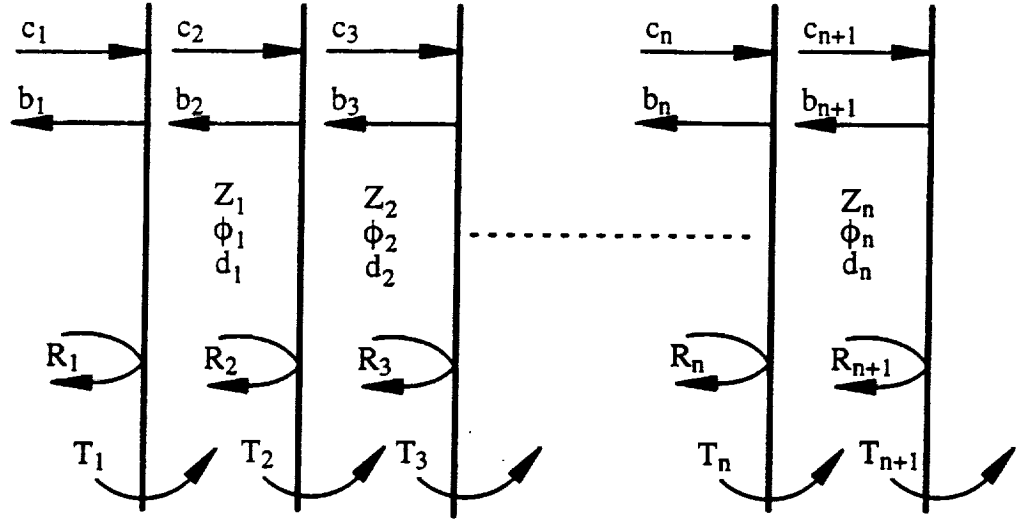


FIGURE X1.4 N Layer Impedance Boundary

When the medium behind the boundary has a characteristic impedance  $Z_{n+1}$  (normalized to free space), we may express the scattering matrix for the entire boundary as

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \prod_{i=1}^{n+1} \frac{1}{T_i} \begin{bmatrix} e^{j\phi_i d_i} & R_i e^{-j\phi_i d_i} \\ R_i e^{j\phi_i d_i} & e^{-j\phi_i d_i} \end{bmatrix} \begin{bmatrix} c_{n+2} \\ b_{n+2} \end{bmatrix} \quad (25.)$$

with  $\phi_{n+1} = 0$ . It is necessary to set  $\phi_{n+1} = 0$  because (as discussed earlier) the appropriate relationship for a simple plane boundary is

$$\begin{bmatrix} c_{n+1} \\ b_{n+1} \end{bmatrix} = \frac{1}{T_{n+1}} \begin{bmatrix} 1 & R_{n+1} \\ R_{n+1} & 1 \end{bmatrix} \begin{bmatrix} c_{n+2} \\ b_{n+2} \end{bmatrix} \quad (26.)$$

The matrix resulting from the  $n+1$  multiplications may be expressed as

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \begin{bmatrix} A_{11} & A_{21} \\ A_{12} & A_{22} \end{bmatrix} \begin{bmatrix} c_{n+2} \\ b_{n+2} \end{bmatrix} \quad (27.)$$

To find the front face reflection coefficient  $R_T$  of the boundary, we simply take the ratio of  $b_1$  to  $c_1$  with  $b_{n+2} = 0$

$$R_T = \left. \frac{b_1}{c_1} \right|_{b_{n+2}=0} = \frac{A_{21}}{A_{11}} \quad (28.)$$

To find the transmission coefficient  $T_T$  for the boundary, we take the ratio of  $c_{n+2}$  to  $c_1$  with  $b_{n+2} = 0$  or

$$T_T = \left. \frac{c_{n+2}}{c_1} \right|_{b_{n+2}=0} = \frac{1}{A_{11}} \quad (29.)$$

To find the front face reflection coefficient with the  $n+1$  medium a perfect conductor, we must alter slightly the preceding formulation as  $T_{n+1} = 0$  which makes the expression indeterminate. This difficulty may be circumvented by considering the boundary as being composed of  $n-1$  layers and the medium behind the boundary as having a normalized characteristic impedance  $Z_c$  where (from transmission line equations)

$$Z_c = Z_n \tanh(j\phi_n d_n) \quad (30.)$$

The calculation proceeds just as before except that now only  $n$  matrices are multiplied and  $\phi_n$  is set equal to 0.

In summary, to compute the reflection and transmission properties of an arbitrary  $n$  layer plane impedance boundary at incidence angle  $\theta$ , we first form and compute

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \prod_{i=1}^{n+1} \frac{1}{T_i} \begin{bmatrix} e^{j\phi_i d_i} & R_i e^{-j\phi_i d_i} \\ R_i e^{j\phi_i d_i} & e^{-j\phi_i d_i} \end{bmatrix} \begin{bmatrix} c_{n+2} \\ b_{n+2} \end{bmatrix} \quad (31.)$$

from which

$$\begin{bmatrix} c_1 \\ b_1 \end{bmatrix} = \begin{bmatrix} A_{11} & A_{21} \\ A_{12} & A_{22} \end{bmatrix} \begin{bmatrix} c_{n+2} \\ b_{n+2} \end{bmatrix} \quad (32.)$$

$$R_T = \frac{A_{21}}{A_{11}} ; \quad T_T = \frac{1}{A_{11}}$$

where we have for TE mode incidence (incident electric vector perpendicular to plane of incidence) and TM or transverse magnetic mode incidence (incident electric vector parallel to plane of incidence)

TE	TM
$\phi_i = \frac{2\pi}{\lambda_0} \sqrt{K_{m_i} K_{e_i} - \sin^2 \theta}$	same

(33.)

$R_i = \frac{Z_i - Z_{i-1}}{Z_i + Z_{i-1}}$	same
---	------

(34.)

$Z_i = \frac{K_{m_i} \cos \theta}{\sqrt{K_{m_i} K_{e_i} - \sin^2 \theta}} \quad (35.)$	$Z_i = \frac{\sqrt{K_{m_i} K_{e_i} - \sin^2 \theta}}{K_{e_i} \cos \theta} \quad (36.)$
--	--



## WAVE-GUIDE CALCULATIONS

It is interesting to note the similarity between the preceding equations describing reflection and transmission of waves in free space and the corresponding equations for wave guides. For instance, if we set the angle of incidence  $\theta$  equal to zero, the preceding equations are identical to those for coaxial wave guide. To make these equations appropriate to the transverse electric  $TE_{n,m}$  and transverse magnetic  $TM_{n,m}$  modes in rectangular wave guide, we must define the angle of incidence  $\theta$  so that

$$\sin \theta = \frac{\lambda_0}{\lambda_{c_{n,m}}}$$

where

$$\lambda_{c_{n,m}} = \text{guide cutoff wavelength for mode } n,m$$

Similarly, for cylindrical wave guide, we define  $\theta$  so that

$$\sin \theta = \frac{\lambda_0 p_{n,m}}{2\pi a} \text{ for } TM_{n,m} \text{ modes}$$

$$\sin \theta = \frac{\lambda_0 p'_{n,m}}{2\pi a} \text{ for } TE_{n,m} \text{ modes}$$

where

$$a = \text{radius of wave guide}$$

$$p_{n,m} = m^{\text{th}} \text{ root of the Bessel function } J_n(k_c a) = 0$$

$$p'_{n,m} = m^{\text{th}} \text{ root of the derivative } \frac{d[J_n(k_c r)]}{dr} \Big|_{r=a} = 0$$

$$k_c = k_{c_{n,m}} = \text{cutoff wave number}$$

$$k_{c_{n,m}} = \frac{p_{n,m}}{a} \text{ for } TM_{n,m} \text{ modes}$$

$$k_{c_{n,m}} = \frac{p'_{n,m}}{a} \text{ for } TE_{n,m} \text{ modes}$$

## COMPUTER PROGRAM

Equations 33. through 36. have been programmed in Fortran IV language for solution by a digital computer. The program computes the power reflection and transmission coefficients for a plane boundary consisting of a finite number of distinct layers. The individual layers may have arbitrary values of complex permittivity and permeability and may be of any thickness. The medium in front of the boundary is assumed to be free space. Calculations are performed for the cases where the medium behind the boundary is either free space or of infinite conductivity (short circuit). The following quantities are computed and printed in the output: RTE, TTE, RTM, TTM, RSCTE, and RSCTM where

RTE = TE mode power reflection coefficient with free space behind boundary

TTE = TE mode power transmission coefficient with free space behind boundary

RTM = TM mode power reflection coefficient with free space behind boundary

TTM = TM mode power transmission coefficient with free space behind boundary

RSCTE = TE mode power reflection coefficient with a short circuit behind boundary

RSCTM = TM mode power reflection coefficient with a short circuit behind boundary

In the printout, all of these quantities are expressed in db (decibels) less than unity. That is

$$R = 20 \log(-|R_T|) \quad (37.)$$

$$T = 20 \log(-|T_T|) \quad (38.)$$

where the logarithm is to base 10 and  $|R_T|$  and  $|T_T|$  are the modulus of the quantities  $R_T$  and  $T_T$  described in Equation 32. If  $R_T$  or  $T_T$  is computed to be 0, then the output  $R$  or  $T$  is set to  $-10^{38}$ .

The output also shows the frequency for which the calculations are made, the incidence angle, and the thickness of individual layers.

The computer program essentially forms and computes Equations 31. and 32. four times for each frequency and aspect angle: TE and TM modes for air and short circuit terminations. The matrices of Equation 31. for short circuit and air backed computation (for the respective modes) differ only in the last two terms as explained previously.

Although only the absolute values of power reflection and transmission coefficients are printed out, the real and imaginary parts could just as well have been chosen. The changes required to select a different output scheme are quite minimal. Also, when the boundary is illuminated on the opposite side, its reflection and transmission properties can be easily examined by simply inverting the resultant 2 by 2 matrices. This latter matrix would then be treated just as the former except that the results would be appropriate to incidence on the opposite side of the boundary.

There is a limit on the number of layers that can be handled. This limit is the parameter MAXLCS, which is based on the values of two other parameters, LAYMX and NEMAX. The

limit is the number of layers contained in a cross section made up of a tenth of the maximum allowable number of elements (NEMAX) in which each element had the maximum allowable number of layers (LAYMX).

### SAMPLE CALCULATION

The results of a three layer problem illuminated by a 1000 MHz wave at incidence angles of 0 and 60 degrees is shown. Each of the layers is 1 cm thick with the following properties:

Front layer	$KE(1) = (1.0, -0.5); KM(1) = (1.0, 0.0)$
Middle layer	$KE(2) = (2.0, -1.0); KM(2) = (1.0, 0.0)$
Back layer	$KE(3) = (5.0, -2.5); KM(3) = (1.0, 0.0)$

The answers for this calculation are as follows:

	Incident Angle	
	<u>0 deg</u>	<u>60 deg</u>
RTE	7.1864	3.9207
TTE	3.2960	6.0755
RTM	7.1864	18.9779
TTM	3.2960	2.7366
RSCTE	1.1104	0.5862
RSCTM	1.1104	3.2879

## SYMBOLS USED IN THE PROGRAM

$A = |R_T|$  of Equation 28. for TE mode  
 $B = |R_T|$  of Equation 28. for TM mode  
 $BETA(I) = \theta$ , incidence angle, degrees  
 $C = |R_T|$  of Equation 28. for TE mode (short circuit calculation)  
 $CE(LI,M) = A_{ij}$  of Equation 32. for TE mode  
 $CES(LI,M) = A_{ij}$  of Equation 32. for TE mode (short circuit calculation)  
 $CM(LI,M) = A_{ij}$  of Equation 32. for TM mode  
 $CMS(LI,M) = A_{ij}$  of Equation 32. for TM mode (short circuit calculation)  
 $D(I) =$  thickness of later I, cm  
 $E = |R_T|$  of Equation 28. for TM mode (short circuit calculation)  
 $EMI =$  ratio of  $b_1$  to  $b_2$  in Equation 20.  
 $EPI =$  ratio of  $c_1$  to  $c_2$  in Equation 20.  
 $F =$  frequency, MHz  
 $K(J) = K_{ei} \times K_{mi}$   
 $KE(I) = K_{ei}$   
 $KM(I) = K_{mi}$   
 $N =$  number of layers  
 $NB =$  number of incidence angles  
 $NF =$  number of frequencies  
 $OLAM =$  free space wavelength in cm  
 $PHI(J) = \phi_i$  (of Equation 33.) times  $d_i$   
 $RE = RTE$   
 $RESC = RSCTE$   
 $RM = RTM$   
 $RMSC = RSCTM$   
 $RTE(J) = R_i$  (Equation 34.) for TE mode  
 $RTESC = R_i$  (Equation 34.) for TE mode (short circuit)  
 $RTM(J) = R_i$  (Equation 34.) for TM mode  
 $RTMSC = R_i$  (Equation 34.) for TM mode (short circuit)  
 $TANN =$  ratio of  $Z_c$  to  $Z_n$  in Equation 30.  
 $TE = TTE$   
 $TEZ = Z_i$  of Equation 35.  
 $TEZSC = Z_i$  of Equation 35. (short circuit)  
 $THETA(L) =$  incidence angle in radians  
 $TM = TTM$   
 $TMZ = Z_i$ , Equation 36.  
 $TMZSC = Z_i$ , Equation 36. (short circuit)

**APPENDIX 3.**  
**ICAN USER MANUAL**



**NASA  
Technical  
Paper  
2515**

**1986**

# **Integrated Composite Analyzer (ICAN)**

*Users and Programmers Manual*

**Pappu L. N. Murthy  
and Christos C. Chamis**

*Lewis Research Center  
Cleveland, Ohio*



National Aeronautics  
and Space Administration

Scientific and Technical  
Information Branch

## Contents

Summary.....	1
Introduction.....	1
Symbols .....	2
Users Manual .....	4
Obtain the Code .....	5
Make it Operational .....	6
Supply the Input Data.....	6
Detailed Description of Input Data .....	8
Title card .....	9
Starting data card.....	9
Booleans .....	9
Ply details card group.....	10
Material system details .....	10
Load cards .....	10
Output .....	11
A Typical IBM Terminal Session .....	15
Programmers Manual .....	16
Main Program .....	16
Subroutine Description .....	18
Subroutine INVA (N, A, C).....	18
Subroutine GACD3 (C) .....	18
Subroutine BLOCK DATA .....	22
Subroutine GPCFD2 (RESF, DISP, PROPC) .....	22
Subroutine COMSA (M).....	25
Subroutine EDGSTR .....	27
Subroutine STRCNF .....	29
Subroutine NUDIFS .....	29
Subroutine MSCBFL (AINF).....	30
Subroutine MCRSTR.....	30
Subroutine MINCOF .....	35
Subroutines AMAXF, AMINF, LOGO, LOGO2.....	36
Subroutine INHYD .....	36
Subroutine FIBMT (C, F, M, VF, VM, VP, KV, IFLAG) .....	37
Subroutine HTM (C, F, M, VF, VM, VV, IFLAG).....	43
Subroutine FLEXX (C) .....	43
Subroutine COMPP (IPFLAG, ISFLAG) .....	44
Subroutines BANKRD and IDGER .....	50
Data base FBMTDATA.BANK .....	51
Appendixes .....	
A-List of Code Identifiers.....	53
B-Input/Output .....	55
C-Resident Data Bank (FBMTDATA.BANK) .....	72
References .....	73



## Summary

This manual describes the use of and relevant equations programmed in a computer code designed to carry out a comprehensive linear analysis of multilayered fiber composites. The analysis contains the essential features required to effectively design structural components made from fiber composites. The program is an outgrowth of two in-house computer codes, MFCA (Multilayered Filamentary Composite Analysis) and INHYD (Intraply Hybrid Composite Design). The inputs to the code are constituent material properties, factors reflecting the fabrication process, and composite geometry. The code performs micromechanics, macromechanics, and laminate analysis, including the hygrothermal response of fiber composites. The code outputs are the various ply and composite properties, composite structural response, and composite stress analysis results with details on failure. The code is in Fortran IV and can be used efficiently as a package in complex structural analysis programs. The input-output format is described extensively through the use of a sample problem. The code manual consists of two parts. The mechanics for using the code are described in the first part, the pertinent equations programmed in the code are described in the second part.

## Introduction

The importance of and need for a multilevel analysis used for designing structural components made of multilayered fiber composites are documented in reference 1. A multilevel analysis, which was efficient in predicting the structural response of multilayered fiber composites (with the constituent material properties, the fabrication process, and the composite geometry known), is also documented in reference 1.

The multilayered analysis presented in reference 1 consists of (1) micromechanical theories for the thermoelastic properties and the stress-level limit of the single ply as functions of constituent material properties and the particular fabrication process, (2) the combined stress-strength criterion for the single ply, and (3) multilayered composite structural response and analysis (macromechanical or laminate analyses), where the interply layer effects are taken into account. A computer code designed to carry out this multilevel analysis, supplemented as noted by references 2 to 10, has been developed at the Lewis Research Center. This code is identified as MFCA for Multilayered Filamentary Composite Analysis (ref. 11).

Intraply hybrid composites are a logical sequel to conventional and interply hybrid composites. Recently, theoretical and experimental investigations have been conducted on the mechanical behavior of intraply hybrids at the Lewis Research Center (refs. 12 to 14). The theoretical methods and equations described in these references, together with those for hygrothermal effects (ref. 15), have been integrated into a computer code for predicting hygral, thermal, and mechanical properties of intraply hybrid composites. This information can then be used in designing these composites. This code is identified as INHYD for Intraply Hybrid Composite Design (refs. 16 and 17).

The present computer code is a synergistic combination of the aforementioned computer programs MFCA and INHYD together with several significant enhancements. The code is referred to as ICAN for Integrated Composite Analyzer. It utilizes the micromechanical design of INHYD and the laminate analysis of MFCA to build a comprehensive analysis capability for structural composites. Additional features unique to ICAN are the following:

- (1) Ply stress-strain influence coefficients
- (2) Microstresses and microstress influence coefficients
- (3) Stress concentration factors around a circular hole
- (4) Calculation of probable delamination locations around a circular hole
- (5) Poisson's ratio mismatch details near a straight free edge
- (6) Free-edge stresses
- (7) Material cards for finite-element analysis using NASTRAN or MARC
- (8) Failure loads, summary based on the maximum stress criterion and laminate failure stresses, and summary based on first-ply failure and fiber breakage criteria
- (9) Transverse shear stresses and normal stresses

In addition to the above, ICAN has its own data base of material properties for commonly used fibers and matrices. The user needs to specify only the coded names for the constituents. The program searches and selects the appropriate properties from its library. Furthermore, input data preparation has been simplified substantially by the introduction of a partial free-field format. The output formats have also been improved significantly to ease user interpretation of the results. These enhancements make ICAN significantly more user friendly than its predecessors. The computer code has been programmed in Fortran IV and has been tested in UNIVAC 1108, IBM 370, and CRAY 1 computers.

Since this report is to serve as a users manual, the code is divided into two parts, the users manual and the programmers manual. The Users Manual describes the mechanics of using the code with respect to program format, input and output, and sample input data sets. The descriptions are extensive enough so that even designers and analysts with little or no programming experience can easily use the code.

The programmers manual gives the various subroutine descriptions and the equations programmed therein, with details on the input and output and the global storage locations. This, along with the listing of the source program, allows the user to make his own modifications to the code as they become appropriate for further enhancements.

The Fortran variables are defined in appendix A. Included is information such as which part of the program of each global variable is generated. Table I provides a summary of details for preparing data cards, and the input data given in table II provide for immediate testing of the code. Properties for a few commonly used fibers and matrix materials are listed in appendix B. Appendix C shows sample input and output data for a specific case.

## Symbols

$A_{cx}$	composite axial stiffness
$A_{cx}^R$	reduced axial stiffness
BIDE	boolean, true if interply effects are included
$C_{cx}$	composite coupling stiffness
$C_{e1}$	string with force variables
$C_{e2}$	string with displacement variables
COMSAT	boolean, true if laminate analysis is wanted
CSANB	boolean, true if membrane and axial symmetry exists
$D_c, D_t$	moisture diffusivity
$D_{cx}$	composite flexural rigidity
$D_{cx}^R$	reduced bending rigidity
$D_v$	displacement vector
$d_f$	filament equivalent diameter
$E_f, E_{cf}$	filament elastic constants
$E_{f11}$ , etc.	fiber normal modulus
$E_p, E_{cp}$	ply elastic constants

$E_{n1}$ , etc.	ply normal modulus
$E_m, E_{cm}$	matrix elastic constants
$E_{m11}$ , etc.	matrix normal modulus
$\epsilon_{m,etc}$	matrix failure strain allowables
$F$	combined stress-failure function
$G_{f12}$ , etc.	fiber shear modulus
$G_{t12}$ , etc.	ply shear modulus
$G_{m12}$ , etc.	matrix shear modulus
$H_j$	interply distortion energy coefficient
$H_{kc}$	array of constituent heat conductivities
$h_c$	composite heat capacity
$i,j$	index, generally ply or interply
$K_{c11,c22,c33}$	composite three-dimensional heat conductivities
$K_{cxx,cyy,cxy}$	composite two-dimensional heat conductivities
$K_{f11}$	fiber heat conductivity
$K_{n1}$	ply heat conductivity
$K_{m11}$	matrix heat conductivity
$k_v$	apparent void volume ratio
$k_f$	actual fiber volume ratio
$k_{ft}$	ply apparent fiber volume ratio
$k_m$	actual matrix volume ratio
$k_{mt}$	ply apparent matrix volume ratio
$k_{vt}$	ply apparent void volume ratio
$L_{sc}$	array of limiting conditions
$M_t$	ply moisture
$M_{cx}$	applied moment
$M_{cM,x}$	hygral moment
$M_{cT,x}$	thermal moment
$m$	load condition index
$N_{cx}$	applied membrane loads
$N_{cM,x}$	hygral force
$N_{cT,x}$	thermal force
$N_f$	number of fibers per end
$N_t$	number of plies
$N_L$	number of load conditions
NONUDF	boolean; true if detailed Poisson's ratio differences chart is to be suppressed
$N_{pc}$	string PROPC length
$N_{pt}$	string PROP length
$P_c$	composite properties array
$P_{cp}$	string PROPC
$P_t$	ply properties array
$P_{tp}$	string PROP main program
$Q_{f,i,p,r,s}$	indices to print out string PROP
$R$	transformation matrix
RINDV	boolean; true if displacements are known

$S_c$	composite failure stress
$S_{n1T}$ , etc.	ply limit failure stresses
$T_l$	ply temperature
$t_l$	ply thickness
$w_{cb}$	composite local curvature changes
$x, y, z$	structural reference axes
$\alpha_c$	composite coefficient of thermal expansion
$\alpha_f$	fiber thermal coefficient of expansion
$\alpha_l$	ply thermal coefficient of expansion
$\alpha_m$	matrix thermal coefficient of expansion
$\beta_c$	moisture expansion coefficients
$\beta_m, \beta_l$	correlation factors for ply thermoelastic properties
$\beta_h$	correlation factor for ply heat conductivity
$\beta_l, \beta_f, \beta_m$	moisture expansion coefficients for ply, fiber, and matrix
$\beta_s$	correlation factor for ply strength
$\beta_v$	matrix strain magnification due to ply strain in the presence of voids
$\delta_f$	interfiber spacing
$\delta_l$	interply layer thickness
$\delta_s$	interfiber spacing
$\epsilon_{cs}$	angle between composite material and structural axes
$\epsilon_{cex}$	reference plane membrane strain
$\epsilon_l$	ply strain
$\theta_g, \theta_k$	angle between ply material and composite axes
$\nu_{f12}$ , etc.	fiber Poisson's ratio
$\nu_{l12}$ , etc.	ply Poisson's ratio
$\nu_{m12}$ , etc.	matrix Poisson's ratio
$\rho_{f,m}$	fiber and matrix weight density
$\rho_{mw}$	density of matrix with moisture
$\sigma_l, \sigma_f, \sigma_m$	ply stresses, fiber stresses, and matrix stresses
1, 2, 3	material reference axes

## Users Manual

The mechanics required to use this code for the analysis of multilayered fiber composites are described in this part of the report. The theory on which the code is based is described in the second part of the report (Programmers Manual).

The physical representations of the constituents used in the code are illustrated in figure 1. This figure shows a complete integration schematic starting with the constituent materials, fiber and matrix. The required input properties and computed properties at various levels are summarized in symbolic form as follows:

(1) Properties required by code as input for a fiber:  $E_{f11,22,33}$ ;  $\nu_{f12,23,13}$ ;  $G_{f12,22,13}$ ;  $\alpha_{f11,22,33}$ ;  $K_{f11,22,33}$ ;  $H_{cf}$ ;  $\rho_f$ ;  $N_f$ ;  $d_f$ ; and  $S_{ff}$ .

(2) Properties required by code as input for a matrix:  $E_{m11,22,33}$ ;  $\nu_{m12,23,13}$ ;  $G_{f12,23,13}$ ;  $\alpha_{m11,22,33}$ ;  $K_{m11,22,33}$ ;  $H_{cm}$ ;  $\rho_m$ ;  $S_{mc}$ ;  $\epsilon_{mpf}$ ;  $\epsilon_{mpc}$ ;  $\epsilon_{mps}$ ; and  $\epsilon_{mpTOR}$ .

(3) Properties required by code as input for a single ply: fiber and matrix properties and ply characteristics  $\beta_c$ ,  $\beta_m$ ,  $\beta_s$ , and  $T_p$ .

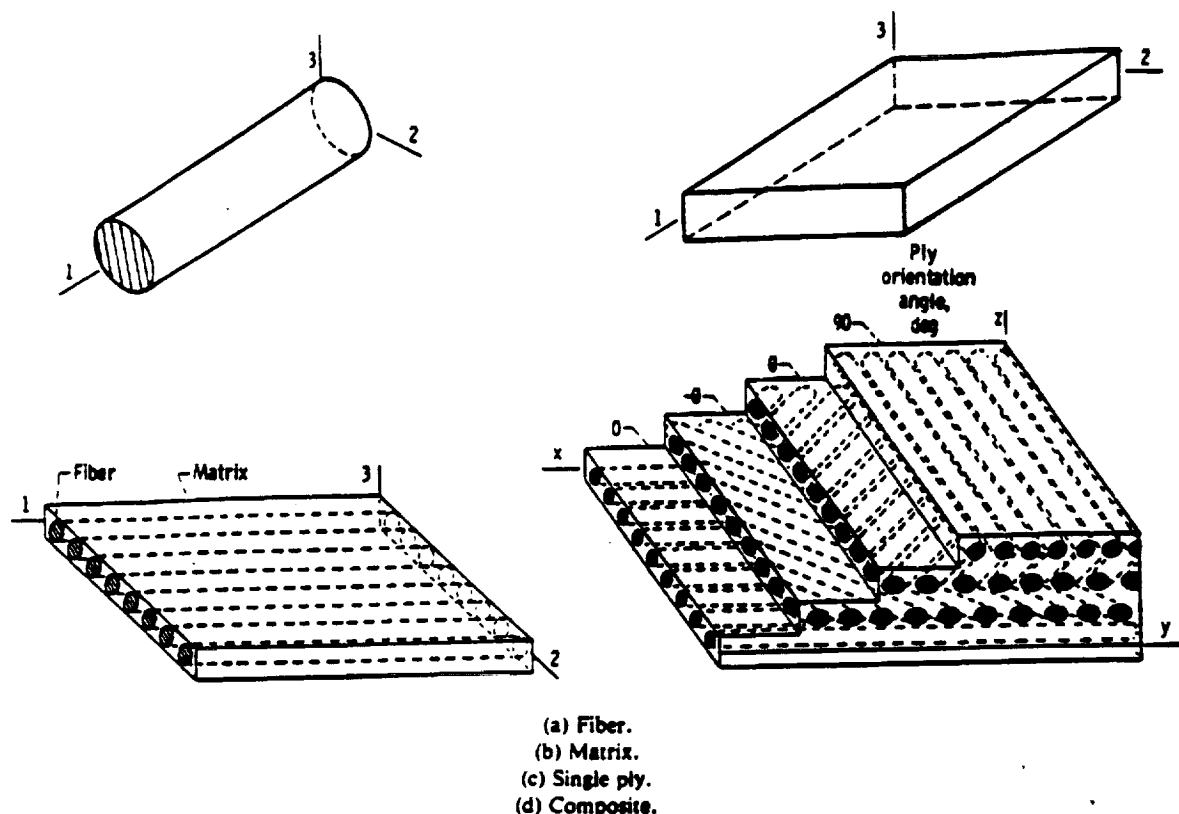


Figure 1.—Schematic of typical multilayered fiber composite and some basic components.

(4) Properties computed by code for single ply:  $E_{11,22,33}$ ;  $\nu_{12,23,13}$ ;  $G_{12,23,13}$ ;  $\alpha_{11,22,33}$ ;  $K_{11,22,33}$ ;  $H_{ci}$ ;  $\rho_i$ ;  $t_i$ ;  $\delta_i$ ;  $S_{11T,11C,22T,22C,12S,23S}$ ;  $K_{12}$ ; and stress analysis factors  $\epsilon_{11,22,12}$ ;  $\sigma_{11,22,12}$ ; and  $1.0 - F(\sigma, S, K_{12})$ .

(5) Properties required by code as input for a composite: ply properties and composite characteristics  $\theta_B$ ,  $H_j$ ,  $K_{12\alpha\beta}$ ,  $N_{cx}$ ,  $M_{cx}$  or  $U_{cx}$  and  $W_{cx}$ .

(6) Output computed by code for a composite:  $\{\epsilon_{cx}\} = [E_c]\{\sigma_c\} + T_t\{\alpha_c\}$ ;  $[E_c]^{-1}$ ;  $K_{cx,yy,xy}$ ;  $H_c$ ;

$$\begin{Bmatrix} N_{cx} \\ M_{cx} \end{Bmatrix} = \begin{bmatrix} A_{cx} & C_{cx} \\ C_{cx} & D_{cx} \end{bmatrix} \begin{Bmatrix} U_{cx} \\ W_{cx} \end{Bmatrix} + \begin{Bmatrix} N_{cx}T_t \\ M_{cx}T_t \end{Bmatrix}$$

and the inverse  $\Delta\phi_{delj}$ .

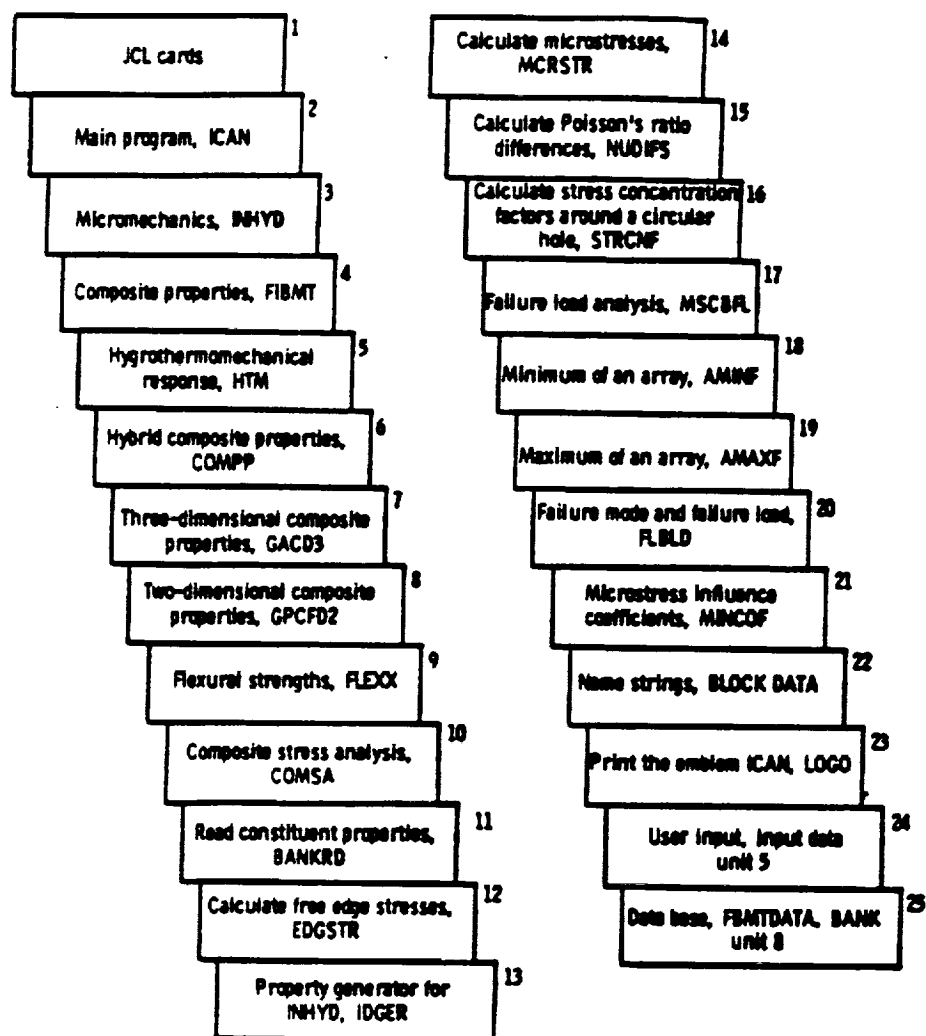
Figure 2(a) shows the subroutines and sequence in the code. The subroutines between the Main program and the input data may be arranged in any desired order. The user should refer to figure 2(b) for the logic flow of the analysis.

The following four steps are required to use the code in the user's computer facility:

- (1) Obtain the code
- (2) Make it operational in the user's computer facility
- (3) Supply the input data
- (4) Interpret the code output results

### Obtain the Code

The code may be obtained in cards. If this is not convenient or possible, the cards can be punched from the compiled listing (contact COSMIC, The University of Georgia, Athens, GA 30602, concerning the availability of this program).



(a) Schematic showing subroutines and sequence of ICAN code.

Figure 2.—Subroutines, sequence, and logic flow of ICAN code.

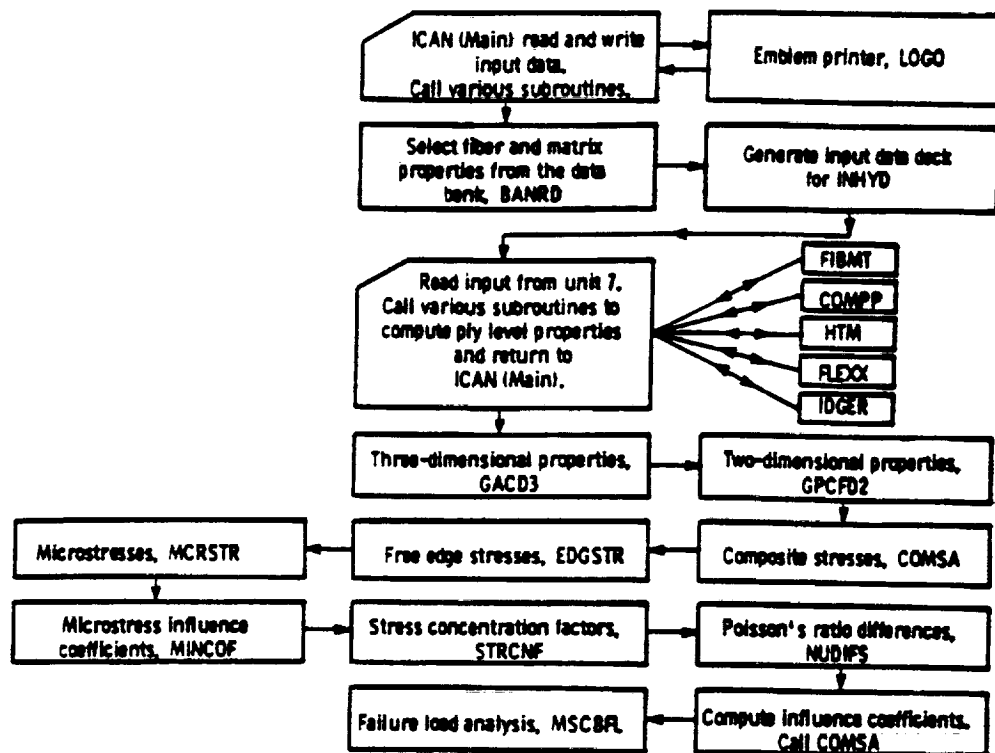
### Make it Operational

A prerequisite to the program is the availability of a Fortran compiler in the user's computer facility. To run the program, certain computer-system-dependent control cards (Job Control Language (JCL) cards) may also be necessary. The computer system personnel should be consulted about these items.

Once the deck of cards has been assembled (except input data) with the proper control cards as shown in figure 2, the user is ready to compile the code in his facility. The compilation will indicate whether any additional modifications are needed. Most modifications will be minor and will usually deal with certain Fortran statements peculiar to each compiler.

### Supply the Input Data

The physical arrangement of the input data cards is illustrated in figure 3. Details for preparing the input data cards are summarized in table I. A detailed description of these cards is given subsequently. A sample for preparing input data for a four-ply symmetric laminate is presented in table II. This laminate has two different material systems. The  $0^\circ$  plies are of AS graphite fiber/intermediate-modulus, low-strength epoxy matrix composite. The  $90^\circ$  plies are made of a hybrid composite. The primary composite is S glass/high-modulus, high-strength epoxy. The secondary composite is AS graphite/intermediate-modulus, high-strength epoxy.



(b) Schematic showing logic flow of ICAN code.

Figure 2.—Concluded.

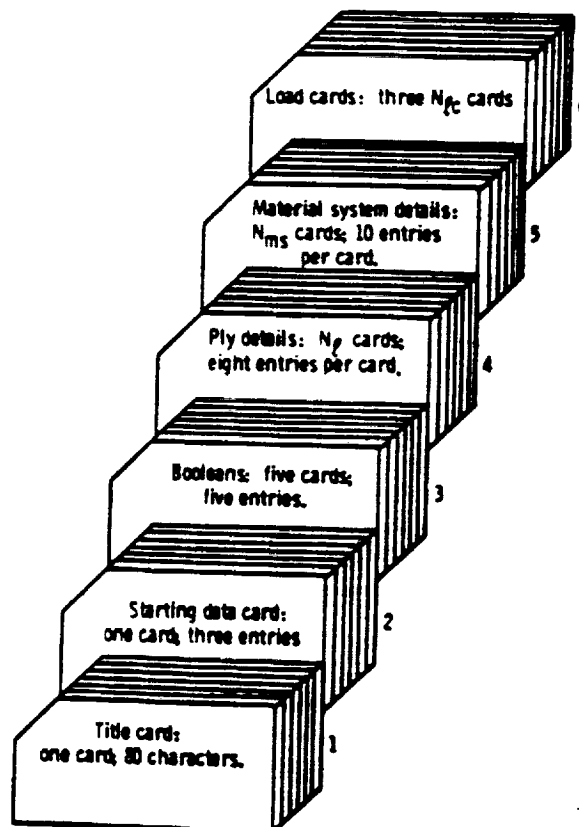


Figure 3.—Physical arrangement of input data cards.

TABLE I.—SUMMARY OF DETAILS FOR PREPARING INPUT DATA CARDS

Card group	Identification	Code symbol	Number of entries	List of entries, sequential order	Card field columns	Format	Comments and engineering units
1	Title card	TITLE	80	Alphabetic characters	1 to 80	(10a8)	
2	STDATA	NL,NLC,NMS	3	$N_L, N_{LC}, N_{MS}$	9 to 36	(a8,31B)	Composite geometries
3	Boolean for input displacement	RINDV	1	_____	1 to 6	_____	T (true) if displays are input; otherwise F (false)
	Boolean for interply layer energy contribution	BIDE	1	_____	1 to 6	_____	T (true) if contributions are desired; otherwise F (false)
	Boolean for membrane and bending symmetry	CSANB	1	_____	1 to 6	(L6)	T (true) if symmetry exists; otherwise F (false)
	Boolean for laminate analysis	COMSAT	1	_____	1 to 6	_____	T (true) if laminate analysis is desired; otherwise F (false)
	Boolean for Poisson ratio chart	NONUDF	1	_____	1 to 6	_____	T (true) if Poisson ratio difference chart is not desired; otherwise F (false)
4	PLY (ply desired)	INP1,IP1,TU,TCU DELM,TETA,THCKNS	7	$i,j, T_{xy}, T_{xz}, \Delta M, \theta_p, k_f$	1 to 64	(a8,21B,5F8.3)	Ply layup and temperature and moisture conditions
5	MATCRD (material system details)	CODES(1,1,J), CODES(1,2,J), VFP,VVP, CODES(2,1,J), CODES(2,2,J), VFS,VVS,VSC	9	Primary composite code names for fiber and matrix, $k_p, k_m$ ; Secondary composite code names for fiber and matrix, $k_{px}, k_{py}, k_z$	1 to 64	(a8,2a4,2E8.2)	Description of material systems to be used
6	PLOAD (loading details)	NX,NY,NXY,THCS	4	$N_x, N_y, N_{xy}$	1 to 32	(a8,7E8.4)	Loading conditions (inplane)
		MX,MY,MXY	3	$\theta_{ex}, M_x, M_y, M_{xy}$	1 to 32	(a8,7E8.4)	Angle of inclination to the structural x-axis
		DMX,DMY PRSSU,PRSSL	4	$dM_x/dx, dM_y/dy, P_{xx}, P_{yy}$	1 to 40	(a8,7E8.4)	Loading conditions (bending) Loading conditions (transverse)

Input data for additional composite systems may be easily prepared. This is done by selecting the desired fiber and matrix from the available materials listed in appendix C using the variable FBMTDATA.BANK and modifying the appropriate entries in the input data sample illustration.

After the input data have been properly assembled (as shown in fig. 3), they are placed in their physical position (fig. 2) and the code is ready to be run.

#### Detailed Description of Input Data

The card group numbers referred to here are given in figure 3 and table I. The sequential order of the entries in each card group is given in table I. Note that most data cards are identified by a mnemonic to indicate the card group in which it belongs in the input data deck. Also, most data cards are divided into fields of eight, with one entry per field being allowed. The mnemonic is entered in



TABLE II.—ICAN SAMPLE INPUT DATASET

FOUR PLY SYMMETRIC LAMINATE. ICAN SAMPLE INPUT DATA.							
STDATA	4	1	2				
T				COMSAT			
F				CSANB			
F				BIDE			
F				RINDV			
T				NONUDF			
PLY	1	1	70.00	70.0	0.0	0.0	0.010
PLY	2	2	70.00	70.0	.0	90.0	.005
PLY	3	2	70.00	70.0	.0	90.0	.005
PLY	4	1	70.00	70.0	.0	.0	.010
MATCRDAS--IMLS	0.55		0.02	AS--IMLS	0.0	0.57	0.03
MATCRDGLAHMMS	.55		.01	AS--IMMS	.4	.57	.01
PLOAD 1000.	0.0		0.0	0.0			NX,MY,NXY,TNCS
PLOAD 0.0	.0		.0				NX,MY,NXY
PLOAD .0	.0						DMX/QX,DMY/QY,PRSS

format A8, and the integers are entered in format I8. The real numbers may be entered anywhere in the appropriate field. The following is a brief description of each card group together with examples taken from table II:

*Title card.*

Any title of length up to 80 characters

FOUR PLY SYMMETRIC LAMINATE. ICAN SAMPLE INPUT DATA.

As shown, any title of length up to 80 characters including blanks may be supplied on this card.

*Starting data card.*

1	8.9	16.17	24.25	a <sub>32</sub>
Mnemonic	N <sub>p</sub>	N <sub>lc</sub>	N <sub>ms</sub>	
STDATA	4	1	2	

This card has a mnemonic STDATA. It contains the overall laminate and loading details. These details are the number of plies N<sub>p</sub>, the number of loading conditions N<sub>lc</sub>, and the number of different material systems N<sub>ms</sub>.

*Booleans.*

1	6.7
Boolean T or F	This space may be used for comments
T	COMSAT
F	RINDV
F	BIDE
F	CSANB
T	NONUDF

A set of booleans, COMSAT, RINDV, BIDE, CSANB, and NONUDF is defined through these cards. These are five cards, one per each logical variable. The format is L6. The variables have the following functions:

(a) COMSAT.—The letter T in the card will direct the program to perform a complete laminate analysis. A letter F would terminate the program at this stage.

(b) RINDV.—The letter T is entered in the card if the displacements are inputs; otherwise, the letter F is entered.

(c) BIDE.—The letter T is entered in the card if the interply layer contributions on the composite are desired; otherwise, the letter F is entered.

(d) CSANB.—The letter T is entered in the card if the composite has both membrane and bending symmetry; otherwise, the letter F is entered.

(e) NONUDF.—The letter T is entered if the detailed Poisson's ratio difference chart is to be suppressed; otherwise, the letter F is entered.

*Ply details card group.*

1	8.9	16.17	24.25	32.33	40.41	48.49	56.57	64
Mnemonic	Ply	Material MID.	$T_u$	$T_{cu}$	$M$	$\theta_f$	$t_f$	
PLY	1	1	70.00	70.0	.0	0.0	.015	
PLY	2	2	70.00	70.0	.0	90.0	.005	
PLY	3	2	70.00	70.0	.0	90.0	.005	
PLY	4	1	70.00	70.0	.0	0.0	.010	

All the cards in this group have the mnemonic PLY. The number of cards is  $N_p$  with eight entries on each card. The first entry is PLY. The second and third entries are identification numbers for the ply and the material system, respectively. The fourth and fifth entries are the use temperature  $T_u$  and the cure temperature  $T_{cu}$ , respectively. The sixth entry is the percentage of moisture  $M$ . The seventh and the eighth entries are the orientation angle  $\theta$  of the ply and the thickness of the ply, respectively. A default value of 0.005 is taken for the thickness if this entry is missing. The material system identification number should be different not only for different composite systems but also for varying use temperature or moisture content from ply to ply.

*Material system details.*

1	8.9	16.17	24.25	32.33	40.41	48.49	56.57	64
Mnemonic	Fiber, matrix	$k_f$	$k_r$	Fiber, matrix	$V_{sc}$	$k_f$	$k_r$	
MATCRD	AS—IMLS	.55	.02	AS—IMLS	0.0	.57	.03	
MATCRD	SGLAHMHS	.55	.01	AS—IMHS	0.4	.57	.01	

All the cards in this group have the mnemonic MATCRD. The number of cards is  $N_{ms}$  with 10 entries in each card. The first entry is MATCRD. The second and the third entries are coded words for fiber and matrix material of the primary composite. The code words are entered in format 2A4. For example, the code for AS-type fiber is AS— and epoxy matrix is EPOX. A dictionary of codes for several fibers and matrices is provided in appendix C. The user may choose any combination of fiber and matrix for a composite system. The fourth and the fifth entries pertain to the details of the primary composite system. They are the primary fiber volume ratio and the primary void volume ratio, respectively. The next two entries refer to the secondary composite system which is applicable for the case of the hybrid composite ply. They should be the same as the second and third entries for standard composite systems. The next entry is the secondary composite system volume ratio. This is zero for the standard composite systems. The last two entries are the fiber volume ratio and the void volume ratio for the secondary composite system. These values are entered when applicable.

*Load cards.*

1	8.9	16.17	24.25	32.33	40
Mnemonic	$N_z$	$N_n$	$N_n$	$T_{ho}$	
PLOAD	1000.	0.0	0.0	0.0	
PLOAD	0.0	0.0	0.0	0.0	
PLOAD	0.0	0.0	0.0	0.0	

All the cards in this group start with the mnemonic PLOAD. There are three cards for each loading condition. Thus, the total number of cards is  $3N_L$ . The first card under each loading condition contains entries  $N_x$ ,  $N_y$ , and  $N_{xy}$  for the membrane loads and  $T_{hcs}$  for the orientation of the loads with respect to the structural axes. Similarly the second card contains the bending resultants  $M_x$ ,  $M_y$ , and  $M_{xy}$ . The last card contains the transverse shear resultants  $DM_x$  and  $DM_y$  and the transverse pressures  $P_u$  and  $P_r$ .

The user input data are read from I/O unit 5. Apart from this, ICAN uses two more units, 7 and 8, for its I/O operations. Unit 8 is used to store the material properties data base. Unit 7 is used as a scratch file by ICAN. These I/O units must be appropriately defined by using the operating system JCL.

### Output

The following items are printed out by the program:

- (1) ICAN logo
- (2) ICAN coordinate systems
- (3) ICAN input data echo
- (4) Input data summary
- (5) Fiber, matrix, and ply level properties of primary and secondary composites
- (6) Composite three-dimensional strain-stress and stress-strain relations about the structural axes; MAT9 card for MSC/NASTRAN solid elements
- (7) Composite properties generated in array PC
- (8) Composite constitutive equations about the structural axes
- (9) Reduced bending and axial stiffnesses
- (10) Data for finite-element analysis
- (11) Displacement-force relations for the current load condition
- (12) Ply hygrothermomechanical properties/response
- (13) Details of Poisson's ratio mismatch among the plies
- (14) Free edge stresses
- (15) Microstresses and microstress influence coefficients for each different composite material system
- (16) Stress concentration factors around a circular hole
- (17) Locations of probable delamination around circular holes
- (18) Ply stress and strain influence coefficients
- (19) Laminate failure load analysis based on the first-ply failure/maximum stress criteria
- (20) Summary of the laminate failure stress analysis based on two alternatives, first-ply failure and fiber breakage

The printout of the input data summary (app. B item 4) shows details regarding composite geometry, constituent specifications, temperature and moisture conditions, and the loading conditions.

The next few pages of the output are generated by the INHYD program package. They show the fiber-matrix properties for the different composite systems and the ply level properties of the composites (app. B, item 5).

The output of the composite three-dimensional strain-stress temperature and moisture relations and composite stress-strain relations about the structural axes are printed under the following headings:

#### (a) 3-D COMPOSITE STRAIN STRESS TEMPERATURE MOISTURE RELATIONS- STRUCTURAL AXES

The matrices  $[E_c]_s^{-1}$ ,  $\{\alpha_c\}_s$ , and  $\{\beta_c\}_s$  in the equation

$$\{\epsilon_c\}_s = [E_c]_s^{-1} \{\sigma_c\}_s - \Delta T_t \{\alpha_c\}_s - M_t \{\beta_c\}_s$$

where  $\Delta T_t = T_t - T_{cu}$

are printed by the subroutine GACD3.

(b) 3-D COMPOSITE STRESS STRAIN RELATIONS-STRUCTURAL AXES  
The matrix  $[E_c]_s$  in the equation

$$\{\sigma_c\}_s = [E_c]_s \{\epsilon_c\}_s$$

is printed out by the subroutine GACD3.

The subscripts in the preceding equations indicate that the relations are written about the structural axes. It is noted that these properties are only local to subroutine GACD3. They can be made global if needed. The properties needed to prepare the MAT9 card of MSC/NASTRAN are printed out next under the heading MAT9 CARD FOR MSC/NASTRAN SOLID ELEMENTS (app. B, item 6).

The output of the composite properties, generated in array PC, are printed under the following heading (app. B, item 7):

COMPOSITE PROPERTIES—VALID ONLY FOR CONSTANT TEMPERATURE THROUGH THICKNESS LINES 1 to 31: 3-D COMPOSITE PROPERTIES ABOUT MATERIAL AXES  
LINES 33 to 62: 2-D COMPOSITE PROPERTIES ABOUT STRUCTURAL AXES

Sixty-two entries are printed under this heading as shown in the following list:

Code name	Notation	Explanation
PC(1)	$\rho_c$	weight density
PC(2)	$t_c$	thickness
PC(3) to PC(11)	$[E_c]$	three-dimensional stress-strain relations about material axes
PC(12) to PC(14)	$\{\alpha_c\}$	three-dimensional coefficients of expansion about material axes
PC(15) to PC(18)	$[K_c], H_c$	three-dimensional heat conductivity and heat capacity along material axes
PC(19) to PC(30)	$E_{c11}, G_{c12}, \nu_{c12}$	three-dimensional constants about material axes
PC(31)	$z_c$	distance to reference plane from bottom of composite
PC(32)	-----	blank
PC(33) to PC(38)	$[E_c]^{-1}$	two-dimensional stress-strain relations about structural axes
PC(39) to PC(47)	$E_{c11}, G_{c12}, \nu_{c12}$	two-dimensional elastic constants along structural axes
PC(48) to PC(54)	$\alpha_c, K_c, H_c$	two-dimensional coefficients of thermal expansion, heat conductivity, and heat capacity along structural axes
PC(55) to PC(58)	$D_c$	moisture diffusivities
PC(59) to PC(62)	$\beta_c$	moisture expansion coefficients

Array PC and its corresponding string and headings are controlled by the formats in subroutine GPCFD2. For nonuniform temperature/moisture, the bending equivalent and the membrane equivalent elastic constants may be obtained by utilizing the reduced bending rigidity matrix and the reduced stiffness matrix which are also regular output of ICAN.

The output for the composite constitutive equations are printed in the following manner (app. B, item 8):

$$\begin{matrix} \text{FORCES} & \text{FORCE DISPLACEMENT} & \text{DISPL} & \text{T-FORCES} & \text{H-FORCES} \\ & \text{RELATIONS} & & & \end{matrix}$$

$$\begin{Bmatrix} \{N_{cx}\} \\ \{M_{cx}\} \end{Bmatrix} = \begin{bmatrix} [A_{cx}] & [C_{cx}] \\ [C_{cx}] & [D_{cx}] \end{bmatrix} \begin{Bmatrix} \{E_{cex}\} \\ \{W_{cbl}\} \end{Bmatrix} - \begin{Bmatrix} \{N_{CT,x}\} \\ \{M_{CT,x}\} \end{Bmatrix} - \begin{Bmatrix} \{N_{CM,x}\} \\ \{M_{CM,x}\} \end{Bmatrix}$$

The elements of matrices  $A_{cx}$ ,  $C_{cx}$ ,  $D_{cx}$ ,  $N_{CT,x}$ ,  $N_{CM,x}$ ,  $M_{CT,x}$ , and  $M_{CM,x}$  are printed out by the subroutine GPCFD2.

The output for the reduced bending rigidities is printed under the heading (app. B, item 9): REDUCED BENDING RIGIDITIES. The elements of  $[D_{cx}^R]$  are printed out as a matrix.

Similarly, the output for the reduced axial stiffness  $[A_{cx}^R]$  is printed out under the heading REDUCED STIFFNESS MATRIX. The corresponding formats for the above two outputs are in subroutine GPCFD2 (app. B, item 10).

The next printout comes from the main program under the heading: SOME USEFUL DATA FOR F.E. ANALYSIS. This information is useful for preparing material data cards for finite element codes NASTRAN and MARC.

The inverse of the constitutive equations is printed out in the following manner (app. B, item 11):

$$\begin{matrix} \text{DISP} & \text{DISPLACEMENT FORCE} & \text{FORCES} \\ & \text{RELATIONS} & \\ \left\{ \begin{matrix} \{ \epsilon_{cxd} \} \\ \{ w_{cb} \} \end{matrix} \right\} = & \left[ \begin{matrix} [A_{cd}] [C_{cd}] \\ [C_{cd}] [D_{cd}] \end{matrix} \right]^{-1} & \left\{ \begin{matrix} [N_{cd}] + [N_{cT,X}] + [N_{cM,X}] \\ [M_{cd}] [M_{cT,X}] [M_{cM,X}] \end{matrix} \right\} \end{matrix}$$

The elements of this inverse are printed out in the subroutine COMSA.

The current values for the loads and corresponding set of ply properties generated in array PL are printed out next (app. B, item 12). The explanations of the 75 entries in the PL property array are given in the following list:

Code name	Notation	Explanation
PL(1,I)	$k_v$	ply void volume ratio
PL(2,I)	$k_{fa}$	ply apparent fiber volume ratio
PL(3,I)	$k_f$	ply actual fiber volume ratio
PL(4,I)	$k_{ma}$	ply apparent matrix volume ratio
PL(5,I)	$k_m$	ply actual matrix volume ratio
PL(6,I)	$\rho_l$	ply weight density
PL(7,I)	$t_l$	ply layer thickness
PL(8,I)	$\delta_l$	ply and interply layer thickness
PL(9,I)	$H_j$	interply layer distortion energy coefficient
PL(10,I)	$z_l$	distance from bottom of composite to ply centroid
PL(11,I)	$z_{cg}$	distance from reference plane to ply centroid
PL(12,I)	$\theta_{cs}$	angle from structural axes to composite material axes (same for all plies) (fig. 2)
PL(13,I)	$\theta_l$	angle from ply material axes to composite material axes (fig. 2)
PL(14,I)	$\theta_s$	angle from ply material axes to composite structural axes (fig. 2)
PL(15,I) to PL(23,I)	$[E_l]^{-1}$	ply stress-strain relations
PL(24,I) to PL(26,I)	$\{\alpha_l\}$	ply thermal coefficients of expansion
PL(27,I) to PL(29,I)	$\{K_l\}$	ply heat conductivities
PL(30,I)	$H_{cl}$	ply heat capacity
PL(34,I) to PL(42,I)	$E_{l11}, \nu_{l12}, G_{l12}$	ply elastic constants
PL(43,I) to PL(48,I)	$D_l$ and $\beta_l$	moisture diffusivities and expansion coefficients
PL(49,I)	$\rho_{del}$	interply delamination factor
PL(50,I)	$T_l$	ply temperature
PL(51,I) to PL(60,I)	$S_{l11T}$ , etc.	ply limiting stresses
PL(61,I)	$K_{l12\alpha\beta}$	coefficient in combined stress-strength criterion
PL(62,I)	-----	combined stress-strength criterion
PL(63,I)	-----	interply delamination criterion
PL(64,I) to PL(69,I)	$\{\epsilon_l\}, \{\sigma_l\}$	ply applied strains and stresses
PL(70,I)	$\Delta\rho_j$	adjacent ply relative rotation
PL(71,I)	-----	Hoffman's failure criterion
PL(72,I)	$M_l$	ply moisture
PL(73,I)	$\sigma_{l13}$	transverse shear stress
PL(74,I)	$\sigma_{l23}$	transverse shear stress
PL(75,I)	$\sigma_{l33}$	thickness stretch stress

The next printout shows Poisson's ratio differences between the plies and the composite (app. B, item 13). They are printed out by the subroutine FESTRE under the heading DETAILS OF POISSON'S RATIO MISMATCH.

The stress peaks near the free edge region are printed out next by the subroutine EDGSTR under the heading (app. C, item 14) FREE EDGE STRESSES.

Item 14 of appendix B shows ply stresses in the structural coordinate system and the through-the-thickness stresses  $\sigma_{xz}$ ,  $\sigma_{zx}$ , and  $\sigma_{yz}$ . The boundary layer decay length is also shown in the table under the heading YDCAY LENGTH. Care must be exercised in interpreting the results. They are based on approximate engineering theories and give good qualitative information regarding the relative magnitudes of the peaks in the individual plies. This printout is suppressed in the case of combined loading.

The microstresses in each ply are printed out next by the subroutine MCRSTR (app. B, item 15(a)). Two regions of interest are considered for the computations, the region between the fibers composed entirely of matrix (A) and the region consisting of fibers as well as matrix (B). The stresses are given a descriptive notation. Thus, SM2AL means stress in matrix along the transverse (2) direction in region A due to a ply stress along the longitudinal direction of the material. Figure 4 shows the definitions for regions A and B. The printout also shows microstresses resulting from moisture and temperature differences if nontrivial  $M_T$  and  $T_T$  are present.

The microstress influence coefficients, stresses due to unit applied stresses in direction 11, 22, 12, 13, and 23 (app. B, item 15(b)); unit temperature difference  $T_T$ ; and unit moisture content  $M_T$  are output from the subroutine MINCOF. These variables are printed out following the microstresses.

Under the heading STRESS CONCENTRATION FACTORS (app. B, item 16) are printed out the factors  $K_{1xx}$ ,  $K_{1yy}$ , and  $K_{1xy}$  which are due to inplane loading around a circular hole at  $5^\circ$  intervals by the subroutine STRCNF. Cumulative stress concentration due to combined loading may be estimated by simple addition of the respective stress concentration factors.

The next output (app. B, item 17) is under the heading POISSON RATIO DIFFERENCES and results from the subroutine NUDIFS. For each ply, the Poisson's ratio differences,  $(\nu_f - \nu_c^{-1})$  and  $(\nu_f - \nu_c)$ , and the products  $K_{1xx}(\nu_f - \nu_c)$ ,  $K_{1yy}(\nu_f - \nu_c)$ , and  $K_{1xy}(\nu_f - \nu_c)$  are printed out at  $5^\circ$  intervals around a circular hole. This is suppressed if the boolean NONUDF is set to TRUE. This item shows the locations of probable delamination for each ply. These are the locations where products such as  $K_{1xx}(\nu_f - \nu_c)$ , for example, are maximum.

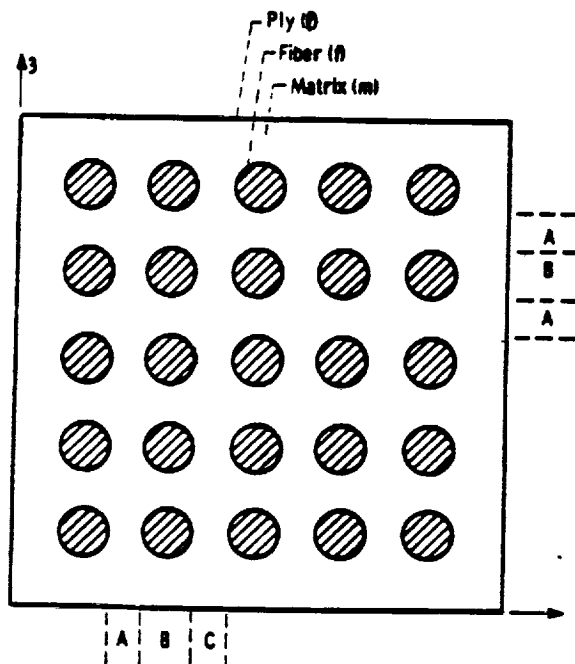


Figure 4.—Definitions of regions for ply microstress calculations.

The next item in appendix B shows the ply stress and strain influence coefficient arrays and ply stress influence coefficient arrays (app. B, item 18). These are computed in the subroutine COMSA and are printed by the main program ICAN. The first table gives the influence coefficients based on unit loads or moments/inch. The second table gives the influence coefficients in terms of unit applied stresses. Explanations of usage of these tables are provided at the end of each table.

The output from the subroutine MSCBFL is printed out next under the heading LAMINATE FAILURE STRESS ANALYSIS (app. B, item 19). The analysis is based on first-ply failure criteria. Results are printed in a tabular form for each ply, and a summary of the analysis is shown in the end (app. B, item 20). The summary shows the critical ply, the failure mode, and the load for each of the applied load types,  $\sigma_{xxT}$ ,  $\sigma_{xxC}$ ,  $\sigma_{yyT}$ ,  $\sigma_{yyC}$ , and  $\sigma_{xyS}$ , respectively. The first table shows results based on first-ply failure, and the second table shows results based on fiber failure by breakage.

### A Typical IBM Terminal Session

To run ICAN, the user must first install and compile the program on his/her computer according to the system to be used. The procedure used on the Lewis Research Center IBM 370 is described in detail here starting from log on. The computer prompt signals are identified with uppercase letters. User entries are in lowercase letters. The following are prerequisites for the user to be able to run ICAN:

- (1) A knowledge of how to compile and store the object module in the public storage space.
- (2) A knowledge of redit or tedit processors so as to be able to create vs datasets of the input data deck. The details of the input data format have already been described in earlier paragraphs.
- (3) A knowledge of commands like rmds, mds, ddef, libdef, print, and erase. These are a few of the commands commonly used in running a program on the IBM 370.

The user is advised to migrate the object deck, the input dataset, and the material property data base so as to conserve his/her permanent storage. The object deck, which is a binary version of the compiled source program, is referred to here as OBJ.ICAN. The data base of material properties is referred to as FBMTDATA.BANK.

The session is started by logging on at the terminal. This is achieved by typing logon, userid, and password. The system replies

```
TSS/370 RELEASE 3.0 PRPZ3 FTF18
SOME MESSAGE          TASKID=OBD7 POOLID=LRCFM-
LOGON AT 11:30 ON 01/15/84
```

The user is now ready for the session. The first phase of the session consists of restoring the necessary data sets to temporary storage. This is achieved by the following commands:

```
rmds obj. ican, aaa
SUCCESSFUL (TEMP) RESTORE OBJ.ICAN AS (AAA)
rmds fbmtdata.bank, ccc
SUCCESSFUL (TEMP) RESTORE FBMTDATA.BANK AS (CCC)
rmds ican.sample.input, bbb
SUCCESSFUL (TEMP) RESTORE ICAN.SAMPLE.INPUT AS (BBB)
```

At this point, the user has all the necessary data sets to run ICAN in his/her temporary storage.

The input/output fortran units that are utilized by ICAN for its various input/output operations need to be defined next. This forms the second phase of the session and is achieved by

```
ddef ft05f001, vs, bbb
ddef ft06f001, vs, icanout.bbb, ret=t
ddef ft08f001, vs, ccc, ret=t
ddef ft07f001, vs, T7, ret=t
```

During these operations, the system usually responds by the minimum prompt, the underscore (\_).

The third phase consists of loading and executing the object deck and printing out the results. This is done by the following commands:

```

libdef lds, aaa
load gpcom$$$
ican
TERMINATED: STOP
print icanout,bbb, prtsp = edit
PRINT BSN = 8835, 1200 LINES

```

The last phase involves cleaning up the user's storage place and is achieved by issuing the commands

```

release lds
release ft
erase aaa
erase bbb
erase ccc
erase t7

```

The user may either logoff or proceed to execute another run for a different set of input data after the preceding set of commands.

## Programmers Manual

A brief description of the main program (or control program) and theoretical equations programmed in the code are presented in this portion of the report. The subroutine descriptions follow the order of execution as shown in the flowchart (fig. 2(b)) rather than the physical sequential order (fig. 2(a)). It is assumed in the following discussion that the user has a working knowledge of computer programming and that he/she is familiar with the terminology appropriate to multilayered composite mechanics.

The assumptions and details leading to the derivation of the equations programmed in the code are not included here. However, they are described in the references cited. It is suggested that the interested user have these references available to him/her.

The information provided in this portion of the code together with the source program listing enables the user to modify, implement, and extend the code according to need.

### Main Program

The main program contains the global variables, the various subroutines, the input data and format, the various program control statements, and the output. These are discussed subsequently. The flowchart of the program is shown in figure 5.

The global variables are given in the following list:

boolean	CSANB, BIDE, RINDV, COMSAT, NONUDF
integers	$N_p, N_{ps}, N_{pc}, N_f, N_k, M, Q_p, Q_s,$ $Q_p, Q_n, Q_f$
real	$\theta_{cp}, \rho_f, \rho_m, d_f, E, \nu, G, f, m, \pi$
real arrays	$K_{xp}, K_{fp}, \theta_k, i(1,1000),$ $P(175,1000), P_c(1,62)$
maximum dimensions	$E_{cp}, E_{cf}, E_{cm}, A_{cp}, C_{cp}, D_{cp}, D_{cp}^R,$ $A_{cp}^R(3,3), \alpha_f, \alpha_m, \alpha_e, N_{cT,X}, M_{dT,X},$ $N_{cM,X}, M_{cM,X}, \epsilon_{czz}, \epsilon_{cbx}(1,3), L_{sc}(1,6),$ $M_{cp}, N_{cp}(3, N_k), D_u(10,6), AINF(6,1000,8),$ $(\lambda_y)_{p,s}, (\lambda_x)_{p,s}, (l_x)_{p,s},$ $(l_y)_{p,s}, (1,1000)$



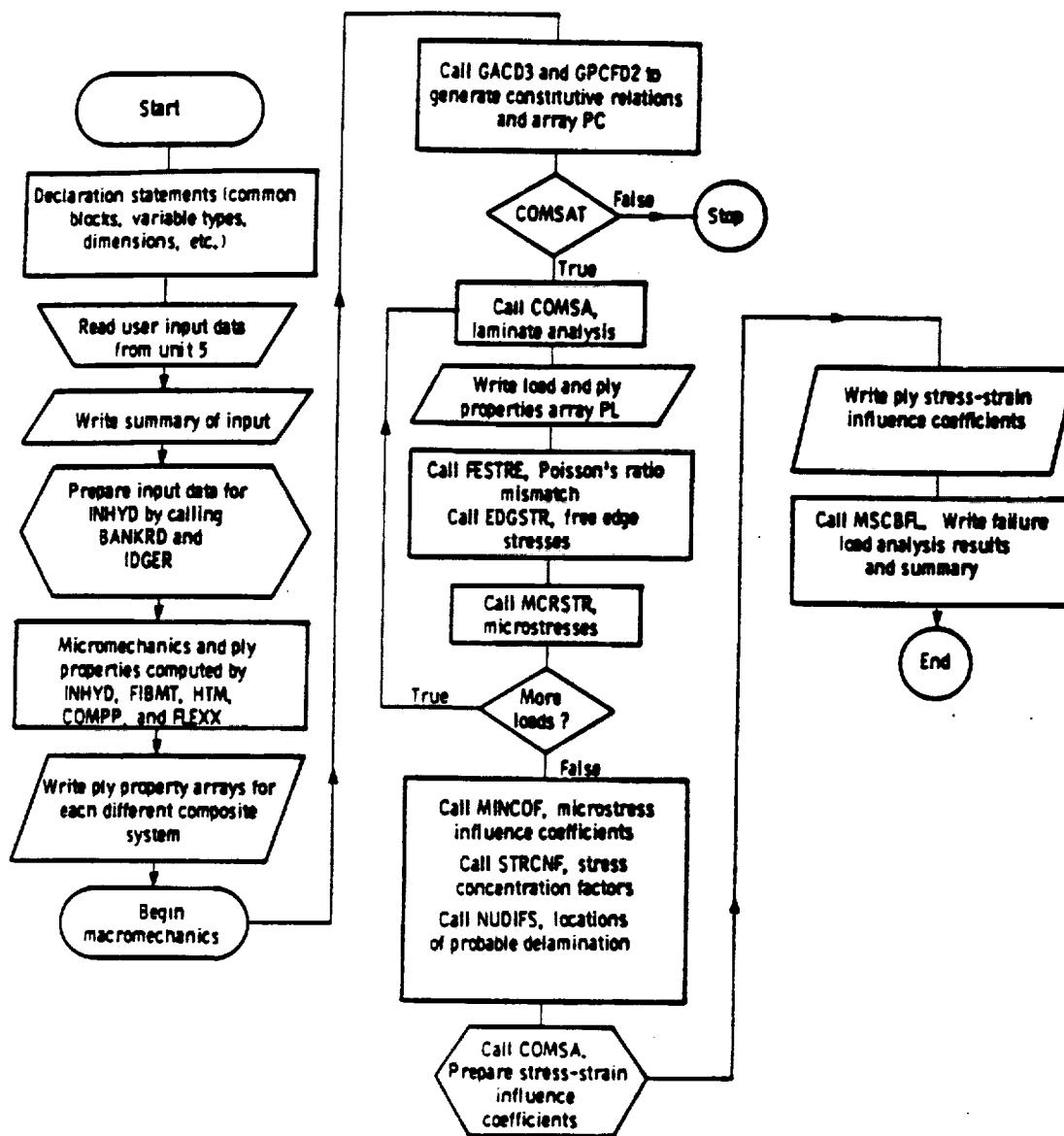


Figure 5.—ICAN program flowchart.

string arrays      title (80 characters) read in.  
                       $P_i$  (eight spaces per field,  $N_{pi}$  fields)  
                       $C_{e1}$  (six spaces per field, six fields)  
                       $C_{e2}$  (six spaces per field, six fields)  
                       $P_{cp}$  (six spaces per field,  $N_{pc}$  fields)  
                      codes C

current dimensions       $N_b, N_{pb}, N_{pc}, N_f, N_{ms}$

real arrays       $K_{rb}, K_{fb}, \theta_k, t_f(1, N_f), P_f(75, N_f)$

current dimensions       $P_c(1, N_{pc}),$   
                       $AINF(6, N_b, 8), \lambda_{y,x,p,s}(1, N_f),$   
                       $t_{x,y,p,s}(1, N_f)$

The subroutines are as follows:

INVA      inverse of an array

GACD3      generates composite three-dimensional elastic and thermal properties and the two-dimensional thermal properties

BLOCK DATA	DISP (String) and RESF (String)
GPCFD2	generates composite two-dimensional elastic constants and constitutive equations
COMSA	generates the ply strain and stress states due to applied loads, checks for ply failure and interply delamination, and generates the ply stress and strain influence coefficients
INHYB	generates ply level properties with the aid of subroutines FIBMT, HTM, COMPP, and FLEXX
BANKRD/IDGER	generates constituent properties by using the data base FBMTDATA.BANK and arranges them in a proper format so as to input to INHYD
FESTRE	computes Poisson's ratio mismatch between the plies and the composite
EDGSTR	computes interlaminar free edge stresses
MCRSTR/MINCOF	generates the microstresses and the corresponding influence coefficients
STRCNF	generates the stress concentration factors around a circular hole
NUDIFS	generates the Poisson's ratio differences within the plies and the probable locations of delamination around the free edge of a circular hole
MSCBFL	performs failure load analysis based on first ply failure/maximum-stress criteria and prints the summary
AMINF	minimum value of an array
AMAXF	maximum value of an array
FLRLD	determines the failure load, failure mode, and the ply location
These subroutines are described in detail in the next section.	
INPUT	title, $N_f$ , $N_{gr}$ , $N_{ms}$ , CSANB, BIDE, RINDV, COMSAT, NONUDF, $t_f$ , $\theta_f$ , $T_f$ , $M_f$ , fiber name, matrix name, $k_{xf}$ , $k_{yf}$ , $k_{zf}$ , $N_{cr}$ , $M_{cr}$ , $DM_{cr}$ , $P_u$ , $P_t$

(For substitution and definition, see appendix A.)

#### Subroutine Description

*Subroutine INVA (N,A,C).*—This subroutine computes the inverse of a square matrix A by Gauss elimination and stores it in array C. The check

$$|A| \neq 0$$

is made and, if satisfied, the program continues; otherwise, the message SINGULAR MATRIX is displayed. The subroutine inputs are N, the matrix order, and the matrix A. The output is

$$A^{-1} - C$$

*Subroutine GACD3(C).*—This subroutine generates the three-dimensional hygrothermoelastic properties of the composite about its structural (x,y,z) and material (1,2,3) axes. The angle  $\theta$  is measured from x of the structural axes system. (See fig. 6.) In figure 6, replace xy etc. by 11 etc. and

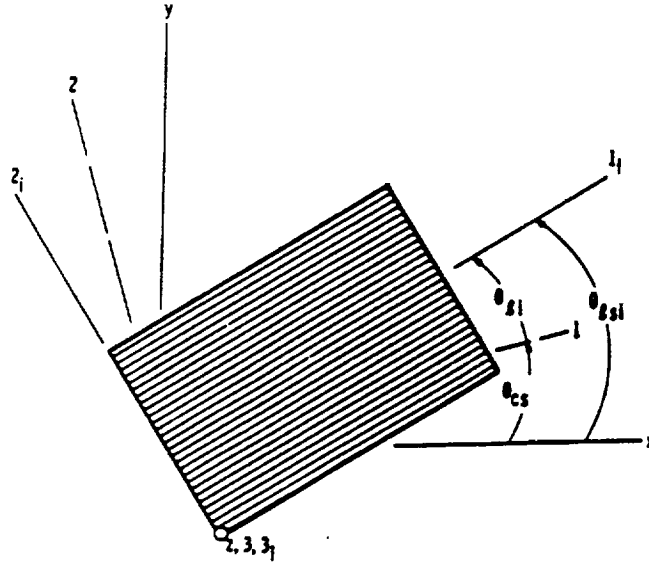


Figure 6.—Ply orientation geometry. Composite structural axes,  $x, y, z$ ; composite material axes,  $1, 2, 3$ ; ply material axes (coincides with fiber direction),  $l, m, n$ .

measure  $\theta$  from the material axes to obtain properties about the material axes. These composite properties are generated from the following equations:

$$[E_c] = \frac{1}{t_c} \left[ \sum_{i=1}^{N_f} (z_{i+1} - z_i) [R_i]^T [E_i] [R_i] + \sum_{j=1}^{N_{f-1}} H_j [S_j] \right]$$

$$\{\alpha_c\} = \frac{1}{t_c} [E_c] \sum_{i=1}^{N_f} (z_{i+1} - z_i) [R_i]^T [E_i] \{\alpha_i\}$$

$$\{\beta_c\} = \frac{1}{t_c} [E_c] \sum_{i=1}^{N_f} (z_{i+1} - z_i) [R_i]^T [E_i] \{\beta_i\}$$

The arrays  $\{\alpha_c\}$ ,  $\{\beta_c\}$ ,  $\{\alpha_i\}$ , and  $\{\beta_i\}$  in the preceding equations are given by

$$\{\alpha_c\} = [\alpha_{cxx} \alpha_{cyy} \alpha_{czz} \alpha_{cyz} \alpha_{czx} \alpha_{cxy}]^T$$

$$\{\beta_c\} = [\beta_{cxx} \beta_{cyy} \beta_{czz} \beta_{cyz} \beta_{czx} \beta_{cxy}]^T$$

and

$$\{\alpha_i\} = [\alpha_{i11} \alpha_{i22} \alpha_{i33} \ 0 \ 0 \ 0]^T$$

$$\{\beta_i\} = [\beta_{i11} \beta_{i22} \beta_{i33} \ 0 \ 0 \ 0]^T$$

For all practical purposes, the two-dimensional thermal coefficients of expansion about the composite structural axes are the same as  $\alpha_{cxx}$ ,  $\alpha_{cyy}$ , and  $\alpha_{cxy}$  in the array  $\{\alpha_c\}$  for the three-dimensional case.

The matrix  $[E_c]^{-1}$  is given by

$$[E_c]^{-1} = \begin{bmatrix} \frac{1}{E_{c11}} & -\frac{\nu_{c21}}{E_{c22}} & \frac{\nu_{c31}}{E_{c33}} & 0 & 0 & 0 \\ -\frac{\nu_{c12}}{E_{c11}} & \frac{1}{E_{c22}} & -\frac{\nu_{c32}}{E_{c33}} & 0 & 0 & 0 \\ -\frac{\nu_{c13}}{E_{c11}} & -\frac{\nu_{c23}}{E_{c22}} & -\frac{1}{E_{c33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{E_{c23}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{E_{c31}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{E_{c12}} \end{bmatrix}$$

Note that for the case of an anisotropic material, the elements (1,6), (2,6), (3,6), and (4,5) and their symmetric parts will not be zero.

The matrices  $[E_g]^{-1}$  and  $[R_g]^{-1}$  are given by

$$[E_g]^{-1} = \begin{bmatrix} \frac{1}{E_{t11}} & -\frac{\nu_{t21}}{E_{t22}} & -\frac{\nu_{t31}}{E_{t33}} & 0 & 0 & 0 \\ -\frac{\nu_{t12}}{E_{t11}} & \frac{1}{E_{t22}} & -\frac{\nu_{t32}}{E_{t33}} & 0 & 0 & 0 \\ -\frac{\nu_{t13}}{E_{t11}} & -\frac{\nu_{t23}}{E_{t22}} & \frac{1}{E_{t33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{E_{t23}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{E_{t31}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{E_{t12}} \end{bmatrix}$$

$$[R_g] = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & 0 & 0 & 0 & \frac{1}{2} \sin 2\theta \\ \sin^2 \theta & \cos^2 \theta & 0 & 0 & 0 & -\frac{1}{2} \sin 2\theta \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos \theta & \sin \theta & 0 \\ 0 & 0 & 0 & -\sin \theta & \cos \theta & 0 \\ -\sin 2\theta & \sin 2\theta & 0 & 0 & 0 & \cos 2\theta \end{bmatrix}_i$$

where  $\theta = \theta_g$  for properties about the composite material and  $\theta = \theta_g + \theta_{cs}$  for properties about the composite structural axes. (See fig. 6.)

The matrix  $[S_{ij}]$  is given by

$$[S_{ij}] = \frac{1}{4} \begin{bmatrix} A^2 & -A^2 & 0 & 0 & 0 & -AB \\ -A^2 & A^2 & 0 & 0 & 0 & AB \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -AB & AB & 0 & 0 & 0 & B^2 \end{bmatrix}_j$$

Here  $A = \sin 2\theta_i - \sin 2\theta_{i-1}$  and  $B = \cos 2\theta_i - \cos 2\theta_{i-1}$  where  $i > 1$  and denotes the ply index.

The angles  $\theta_i$  and  $\theta_{i-1}$  (fig. 6) are given by

$$\theta_i = \theta_g + \theta_{cs}$$

$$\theta_{i-1} = \theta_{g-1} + \theta_{cs}$$

The composite heat capacity is the same for both the two- and the three-dimensional cases. It is given by

$$h_c = \frac{1}{t_c} \sum_{i=1}^{N_f} h_g t_g$$

and  $t_c$  is given by

$$t_c = \sum_{i=1}^{N_f} t_g$$

The composite three-dimensional heat conductivities along the composite material axes, assuming an orthotropic composite, are given by

$$K_{c11} = \frac{1}{t_c} \sum_{i=1}^{N_f} t_g (K_{f11} \cos^2 \theta_i + K_{f22} \sin^2 \theta_i)_i$$

$$K_{c22} = \frac{1}{t_c} \sum_{i=1}^{N_f} t_g (K_{f11} \sin^2 \theta_i + K_{f22} \cos^2 \theta_i)_i$$

$$\frac{1}{K_{c33}} = \frac{1}{t_c} \sum_{i=1}^{N_f} \left( \frac{t_f}{K_{f33}} \right)_i$$

The angle  $\theta_i$  is measured from the material axes (fig. 6)

The composite two-dimensional heat conductivities along the composite structural axes are given by (see ref. 9 for the transformation equations)

$$K_{c\alpha\alpha} = \frac{1}{t_c} \sum_{i=1}^{N_f} t_g (K_{f11} \cos^2 \theta + K_{f22} \sin^2 \theta)_i$$

$$K_{c\gamma\gamma} = \frac{1}{t_c} \sum_{i=1}^{N_f} t_g (K_{f11} \sin^2 \theta + K_{f22} \cos^2 \theta)_i$$

$$K_{c\gamma\alpha} = K_{c\alpha\gamma} = \frac{1}{t_c} \sum_{i=1}^{N_f} t_g (K_{f22} - K_{f11})_i \sin 2\theta_i$$

$$K_{c\alpha\alpha} = K_{c33}$$

The angle  $\theta$  in the last set of equations is measured from the composite structural axes and is equal to  $\theta_{\alpha\alpha} + \theta_f$ . The inputs to the subroutine are  $N_f$ ,  $z_{g+1}$ ,  $z_g$ ,  $\theta_{\alpha\alpha}$ ,  $\theta_{1f}$ ,  $[E_f]$ ,  $H_f$ ,  $[\alpha_g]$ ,  $h_g$ , and  $[K_g]$ , which are all global. The variable  $N_f$  is input data. The remaining quantities are either generated or are transferred from information stored in PL(11,1), PL(13,1), PL(15,1-23,1), PL(8,1), PL(24,1) to PL(26,1), PL(30,1), PL(27,1), and PL(29,1). The outputs are  $t_c$  and the arrays are  $[E_c]^{-1}$ ,  $[\alpha_c]$ ,  $[E_c]$ ,  $h_c$ , and  $[K_c]$ . The composite thickness  $t_c$  is stored in PC(2). The arrays  $[E_c]^{-1}$ ,  $[\alpha_c]$ , and  $[E_c]$  for both composite material and structural axes are printed out under the headings 3-D COMPOSITE STRAIN STRESS TEMPERATURE MOISTURE RELATIONS-STRUCTURAL AXES and 3-D COMPOSITE STRESS STRAIN RELATIONS-STRUCTURAL AXES.

The composite material axes properties  $[E_c]$  and  $[\alpha_c]$  are stored in PC(3) to PC(14) as global variables. The corresponding moduli are stored in PC(19) to PC(30). The three-dimensional heat conductivities and heat capacity along the material axes are stored in PC(15) to PC(18). The two-dimensional thermal coefficients of expansion along the structural axes are stored in PC(48) to PC(50). The two-dimensional heat conductivities and heat capacity along the structural axes are stored in PC(51) to PC(54). Note that the heat capacity is a scalar quantity and is independent of the reference axes. Therefore, PC(54) equals PC(18). The moisture diffusivities and expansion coefficients are stored in entries PC(55) to PC(62).

**Subroutine BLOCK DATA.**—In this block, the strings  $C_{e1}$  and  $C_{e2}$ , which are printed out with the composite constitutive equations, are defined. The string  $C_{e1}$  contains the resultant force notation  $N_{cx}$ ,  $N_{cy}$ ,  $N_{cxy}$ ,  $M_{cx}$ ,  $M_{cy}$ , and  $M_{cxy}$ . The string  $C_{e2}$  contains the notation for the corresponding displacements.

**Subroutine GPCFD2 (RESF, DISP, PROPC).**—This subroutine generates the required section properties and the force-deformation temperature-moisture relations for a two-dimensional

multilayered composite. It also generates the plane-stress elastic constants for the composite. The force-deformation temperature-moisture relations generated in this procedure are defined in the following equation:

$$\begin{Bmatrix} [N_{cd}] \\ [M_{cd}] \end{Bmatrix} = \begin{bmatrix} [A_{cd}] & [C_{cd}] \\ [C_{cd}] & [D_{cd}] \end{bmatrix} \begin{Bmatrix} \epsilon_{cd} \\ w_{cd} \end{Bmatrix} - \begin{Bmatrix} [N_{cT,d}] \\ [M_{cT,d}] \end{Bmatrix} - \begin{Bmatrix} [N_{cM,d}] \\ [M_{cM,d}] \end{Bmatrix}$$

The generic equations for the elements in the arrays  $[A_{cd}]$ ,  $[C_{cd}]$ ,  $[D_{cd}]$ ,  $[N_{cT,d}]$ ,  $[M_{cT,d}]$ ,  $[N_{cM,d}]$ , and  $[M_{cM,d}]$  are

$$\begin{aligned} [A_{cd}] &= \sum_{i=1}^{N_l} (z_{i+1} - z_i) [R_i]^T [E_i]^{-1} [R_i] + \sum_{j=1}^{N_l-1} H_j [S_j] \\ [C_{cd}] &= \frac{1}{2} \sum_{i=1}^{N_l} (z_{i+1}^2 - z_i^2) [R_i]^T [E_i]^{-1} [R_i] + \sum_{j=1}^{N_l-1} z_{rpj} H_j [S_j] \\ [D_{cd}] &= \frac{1}{3} \sum_{i=1}^{N_l} (z_{i+1}^3 - z_i^3) [R_i]^T [E_i]^{-1} [R_i] + \frac{1}{2} \sum_{j=1}^{N_l-1} z_{rpj}^2 H_j [S_j] \\ [N_{cT,d}] &= \sum_{i=1}^{N_l} \Delta T_i (z_{i+1} - z_i) [R_i] [E_i]^{-1} [\alpha_i] \\ [N_{cM,d}] &= \sum_{i=1}^{N_l} M_i (z_{i+1} - z_i) [R_i] [E_i]^{-1} [\beta_i] \\ [M_{cT,d}] &= \frac{1}{2} \sum_{i=1}^{N_l} \Delta T_i (z_{i+1}^2 - z_i^2) [R_i]^T [E_i]^{-1} [\alpha_i] \\ [M_{cM,d}] &= \frac{1}{2} \sum_{i=1}^{N_l} M_i (z_{i+1}^2 - z_i^2) [R_i]^T [E_i]^{-1} [\beta_i] \end{aligned}$$

where  $\Delta T_i = T_i - T_{cui}$

The arrays  $[\alpha_i]$ ,  $[\beta_i]$ ,  $[R_i]$ ,  $[E_i]$ , and  $[S_j]$  are

$$[\alpha_i] = [\alpha_{11} \quad \alpha_{22} \quad 0]^T$$

$$[\beta_i] = [\beta_{11} \quad \beta_{22} \quad 0]^T$$

$$[R_i] = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & \frac{1}{2} \sin 2\theta \\ \sin^2 \theta & \cos^2 \theta & -\frac{1}{2} \sin 2\theta \\ -\sin 2\theta & \sin 2\theta & \cos 2\theta \end{bmatrix}$$

$$[E_g] = \begin{bmatrix} \frac{1}{E_{11}} & -\frac{\nu_{12}}{E_{11}} & 0 \\ -\frac{\nu_{12}}{E_{11}} & \frac{1}{E_{22}} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix}$$

$$S_{j22} = S_{j11} = \frac{1}{4} (\sin 2\theta_i - \sin 2\theta_{i-1})^2$$

$$S_{j21} = S_{j12} = -S_{j11}$$

$$S_{j32} = S_{j23} = \frac{1}{4} (\sin 2\theta_i - \sin 2\theta_{i-1})(\cos 2\theta_i - \cos 2\theta_{i-1})$$

$$S_{j31} = S_{j13} = -S_{j23}$$

$$S_{j33} = \frac{1}{4} (\cos 2\theta_i - \cos 2\theta_{i-1})^2$$

Here  $\theta_i$  equals the  $\theta_{cx} + \theta_i$  (fig. 6). The reduced bending rigidities (ref. 6) are generated in this procedure according to the equation

$$D_{cx}^R = [D_{cx} - C_{cx} A_{cx}^{-1} C_{cx}]$$

The reduced axial stiffnesses are generated in the procedure according to the equation

$$A_{cx}^R = [A_{cx} - C_{cx} D_{cx}^{-1} C_{cx}]$$

The two-dimensional composite elastic constants are generated from the following equation (assuming  $T_g = T_i$  for  $i = 1$  to  $N_f$  and  $M_g = M_i$  for  $i = 1$  to  $N_\theta$ ):

$$[E_{cx}]^{-1} = \frac{1}{t_c} \left\langle \sum_{i=1}^{N_f} (z_{g+1} - z_g) [R_g]^T [E_g]^{-1} [R_g] + \sum_{j=1}^{N_\theta} H_j [S_j] \right\rangle$$

where

$$t_c = \sum_{i=1}^{N_f} t_g$$

The inputs to this subroutine are  $t_g$ ,  $T_g$ ,  $M_g$ ,  $\theta_i$  (relative to composite structural axes),  $H_j$ , and the ply elastic constants. These quantities are global and are located, respectively, in PL(7,1), PL(50,1), PL(72,1), PL(14,1), PL(9,1), and PL(31,1) to PL(42,1). The arrays  $[R_g]^T$ ,  $[E_g]^{-1}$ ,  $[R_g]$ , and  $[S_j]$  and the dimensions  $z_g$  are generated within this subroutine.

The outputs are the force-deformation temperature-moisture relations, which are stored in the global arrays  $ACX = A_{cx}$ ,  $RAC = A_{cx}^R$ ,  $CPC = C_{cx}$ ,  $FLX = D_{cx}$ ,  $RDC = D_{cx}^R$ ,  $NSDT = N_{cT,\mu}$ ,  $MSDT = M_{cT,\mu}$ ,  $NSDH = N_{cM,\mu}$ , and  $MSDH = M_{cM,\mu}$ . These are printed out under the heading



FORCES FORCE DISPLACEMENT RELATIONS DISPL T-FORCES H-FORCES. The reduced bending rigidities are printed out under the heading REDUCED BENDING RIGIDITIES. The reduced axial stiffnesses are printed out under the heading REDUCED STIFFNESS MATRIX. The inverse of the constitutive equations

$$\begin{bmatrix} [A_{cx}] & [C_{cx}] \\ [C_{cx}] & [D_{cx}] \end{bmatrix}^{-1}$$

are printed out under the heading DISP DISPLACEMENT FORCE RELATIONS FORCES. The distances  $z_c$ ,  $z_g$ , and  $z_g$  are stored in PC(31,I), PL(10,I), and PL(11,I), respectively. The two-dimensional composite stress-strain relations are stored in PC(33) to PC(38), and the two-dimensional composite moduli and Poisson's ratios are stored in PC(39) to PC(47). The two-dimensional thermal properties are stored in PC(48) to PC(54), as is described in the section subroutine GACD3.

**Subroutine COMSA (M).**—In this subroutine the stress and strain states of each ply are computed given the edge membrane forces, the ply temperature, and the changes in curvature. In addition, two-ply, combined stress-strength criteria and the interply delamination criterion are generated. Also generated are the ply stress-strain influence coefficients. The equations programmed for the  $k$ th strain and stress states are

$$\begin{aligned} \{\epsilon_g\} &= [R_g][A_{cx}]^{-1} \{ [N_{cx}] + [N_{cT,x}] + [N_{cM,x}] + [C_{cx}]\{w_{cbx}\} \} - z[R_g]\{w_{cbx}\} \\ \{\sigma_g\} &= [E_g]^{-1} [R_g][A_{cx}]^{-1} \{ [N_{cx}] + [N_{cT,x}] + [N_{cM,x}] + [C_{cx}]\{w_{cbx}\} \} \\ &\quad - [E_g]^{-1} \{ T_g\{\alpha_g\} + M_g\{\beta_g\} + z[R_g]\{w_{cbx}\} \} \end{aligned}$$

The reference plane strains  $\epsilon_{cex}$  and the curvature changes are computed from

$$\begin{Bmatrix} \{\epsilon_{cex}\} \\ \{w_{cbx}\} \end{Bmatrix} = \begin{bmatrix} [A_{cx}] & [C_{cx}] \\ [C_{cx}] & [D_{cx}] \end{bmatrix}^{-1} \left\{ \begin{Bmatrix} [N_{cx}] \\ [M_{cx}] \end{Bmatrix} + \begin{Bmatrix} [N_{cT,x}] \\ [M_{cT,x}] \end{Bmatrix} + \begin{Bmatrix} [N_{cM,x}] \\ [M_{cM,x}] \end{Bmatrix} \right\}$$

when either the membrane force or the moments or both are given.

The strains are generated locally in EPSL and SIGL, respectively, and are stored in PL(64,I) to PL(69,I). The matrices  $[R_g]$  and  $[E_g]$  are generated locally from information transferred from PL(14,I) and PL(31,I) to PL(42,I). The distance  $z_g$ , the ply temperature  $T_g$ , and the ply moisture  $M_g$  are transferred from PL(11,I), PL(50,I), and PL(72,I), respectively. The remaining matrices are

$A_{cx}$  — ACX  
 $C_{cx}$  — CPC  
 $N_{cT,x}$  — NSDT  
 $N_{cM,x}$  — NSDH  
 $N_{cx}$  — NSB<sub>m</sub>  
 $M_{cT,x}$  — MSDT  
 $M_{cM,x}$  — MSDH  
 $M_{cx}$  — MSB<sub>m</sub>

and  $w_{cbx}$  — WXX<sub>m</sub> (local curvature from bending analysis), where  $m$  denotes the load condition.

It is important to note that the stress analysis in the coded form also handles the case where both the reference plane membrane strains and the local curvatures are given. In this case the ply strains are given by

$$\{\epsilon_{cxi}\} = \{\epsilon_{cxi}\} - z\{w_{cxi}\}$$

where  $\{\epsilon_{cxi}\}$  is the  $i$ th ply strain along the structural axis,  $\{\epsilon_{cxi}\}$  is the reference plane membrane strain,  $z$  is the distance from the reference plane to the centroid of the  $i$ th ply, and  $\{w_{cxi}\}$  is the local curvature. These variables are read in the array  $D_{cm}$ , where  $m$  denotes the load condition.

The corresponding  $i$ th ply stresses are given by

$$\{\sigma_i\} = [E_i]^{-1} \{ [R_i] \{\epsilon_{cxi}\} - \Delta T_i \{\alpha_i\} - M_i \{\beta_i\} \}$$

$$\Delta T_i = T_i - T_{cui}$$

where  $\{\sigma_i\}$  is the  $i$ th ply stress along the material axes,  $[E_i]$  is the  $i$ th ply elastic constant about the material axes,  $[R_i]$  is the transformation matrix of the  $i$ th ply,  $\{\epsilon_{cxi}\}$  is the  $i$ th ply strain along the structural axes as given by a previous equation,  $T_i$  is the temperature of the  $i$ th ply,  $T_{cui}$  is the cure temperature of the  $i$ th ply,  $\{\alpha_i\}$  is the thermal coefficient of expansion of the  $i$ th ply along the material axes,  $M_i$  is the moisture content of the  $i$ th ply, and  $\{\beta_i\}$  is the moisture expansion coefficient of the  $i$ th ply along the material axes.

The displacement force relations are printed out in the following format:

DISPLACEMENT	DISPLACEMENT FORCE RELATIONS	FORCES
$\begin{Bmatrix} \{U_{cx}\} \\ \{W_{cx}\} \end{Bmatrix}$	$\begin{bmatrix} [A_{cx}] & [C_{cx}] \\ [C_{cx}] & [D_{cx}] \end{bmatrix}^{-1}$	$\begin{Bmatrix} \{N_{cx}\} \\ \{M_{cx}\} \end{Bmatrix}$

Two similar sets are printed out. In the first set, the displacement and force vectors are in symbolic form. In the second set, the displacement and force vectors have their numerical values. (See outputs of trial cases, app. B.)

The failure criterion may be determined by either of the following methods:

(1) Modified distortion energy

$$F = 1 - \left[ \left( \frac{\sigma_{11\alpha}}{S_{11\alpha}} \right)^2 + \left( \frac{\sigma_{22\alpha}}{S_{22\alpha}} \right)^2 - K_{112\alpha} \frac{\sigma_{11\alpha}}{S_{11\alpha}} \frac{\sigma_{22\alpha}}{S_{22\alpha}} + \left( \frac{\sigma_{12\alpha}}{S_{12\alpha}} \right)^2 \right] - PL(62,1)$$

The parameters  $\alpha$  and  $\beta$  are specified as follows:

$$\alpha = \begin{cases} T & \sigma_{11} \geq 0 \\ C & \sigma_{11} < 0 \end{cases}$$

$$\beta = \begin{cases} T & \sigma_{22} \geq 0 \\ C & \sigma_{22} < 0 \end{cases}$$

$$S_{11\alpha} = \begin{cases} S_{11T} & \alpha = T \\ \min(S_{11C}, S_{11CD}) & \alpha = C \end{cases}$$

$$S_{22\alpha} = \begin{cases} S_{22T} & \beta = T \\ S_{22C} & \beta = C \end{cases}$$

$$K_{i12\alpha\beta} = K'_{i12\alpha\beta} \frac{(1 + 4\nu_{i12} - \nu_{i13})E_{i22} + (1 - \nu_{i23})E_{i11}}{[E_{i11}E_{i22}(2 + \nu_{i12} + \nu_{i13})(2 + \nu_{i21} + \nu_{i23})]^{1/2}}$$

$$K'_{i12\alpha\beta} = \begin{cases} \text{BET}(1, 7) & \alpha, \beta = T \\ \text{BET}(2, 7) & \alpha = C, \beta = T \\ \text{BET}(1, 8) & \alpha = T, \beta = C \\ \text{BET}(2, 8) & \alpha, \beta = C \end{cases}$$

The multiplier of  $K'_{i12\alpha\beta}$  was generated in the main program and is stored in PL(61,1). The constant  $K'_{i12\alpha\beta}$  constitute theory-experiment correlation factors. These are set as unity in COMSA. However, the user can modify the correlation factors if he/she wishes, by redefining the matrix BET in the subroutine COMSA.

(2) Hoffman's criterion (ref. 9)

$$S_{i11C} = \min(S_{i11C}, S_{i11CD})$$

$$F = 1 - \left[ \frac{\sigma_{i11}^2 - \sigma_{i11}\sigma_{i22}}{S_{i11C}S_{i11T}} + \frac{\sigma_{i22}^2}{S_{i22C}S_{i22T}} + \frac{S_{i11C} - S_{i11T}}{S_{i11C}S_{i11T}}\sigma_{i11} + \frac{S_{i22C} - S_{i22T}}{S_{i22C}S_{i22T}}\sigma_{i22} + \frac{\sigma_{i12}^2}{S_{i12S}^2} \right] - \text{PL}(71,1)$$

$F > 0$  no failure

$F = 0$  incipient failure

$F < 0$  failure

The interply delamination criterion for the  $j$ th interply layer at the  $m$ th load condition is governed by

$$\left[ 1 - \left( \frac{|\Delta\phi|}{\Delta\phi_{\text{del}}} \right) \right]_j - \text{PL}(63,1) \quad \text{when } i > 1$$

$$\Delta\phi_j = \frac{1}{2}(\epsilon_{cyy} - \epsilon_{c\alpha\alpha})(\sin 2\theta_i - \sin 2\theta_{i-1}) + \frac{1}{2}\epsilon_{cxy}(\cos 2\theta_i - \cos 2\theta_{i-1})(\epsilon_{c\alpha\alpha}) = [A_{c\alpha}]^{-1} \{ [N_{c\alpha}] + [N_{cT,\alpha}] + [N_{cM,\alpha}] + [C_{c\alpha}](w_{c\alpha\alpha}) \}$$

or by the displacement force equation described previously.

The inputs to the subroutine are the ply angle measured from the structural axes ( $\theta_i$ , from PL(14,1)); the distance from the reference plane to the centroid of the ply ( $z_g$ , from PL(11,1)); the ply temperature ( $T_g$ , from PL(50,1)); the interply delamination limit ( $\Delta\phi_{\text{del},j}$ , from PL(60,1)); the ply thermoelastic properties stored in PL(24 to 26,1) and PL(31 to 42,1); the ply extensional and coupling rigidities,  $A_{c\alpha} = ACX$  and  $C_{c\alpha} = CPC$ ; the local curvatures  $w_{c\alpha\alpha} = WXX$ ; the adjustment constants  $K'_{i12TT} = \text{BET}(1, 7)$ ,  $K'_{i12CT} = \text{BET}(2, 7)$ ,  $K'_{i12TC} = \text{BET}(1, 8)$ , and  $K'_{i12CC} = \text{BET}(2, 8)$ ; and the load conditions  $N_{c\alpha} = \text{NBS}(m)$ .

The subroutine outputs are the modified distortion energy PL(62,1), Hoffman's criterion PL(71,1), the interply delamination criterion PL(63,1), and the adjacent ply relative rotation ( $\Delta\phi_j$ , from PL(70,1)).

**Subroutine EDGSTR.**—This subroutine computes the interlaminar stresses  $\sigma_{c\alpha}$ ,  $\sigma_{c\beta}$ , and  $\sigma_{c\gamma}$  near a straight free edge region of a finite width, infinitely long plate under uniform extension. The equations used are based on an approximate formulation analogous to that in reference 18. The calculations are performed in two parts. The first part consists of computations of decay lengths for

the interlaminar stresses. The decay length is a measure of a free edge region in which the interlaminar stresses may be significant. This is achieved in the main program. The second part uses this information to compute the interlaminar stresses in the subroutine EDGSTR. The pertinent equations are discussed in the following paragraphs. Note that in the case of hybrid composite plies, the calculations are repeated not only for the primary composite but also for the secondary composite by using the appropriate ply constituent properties. The primary and the secondary composites are distinguished by using the letters P and S, respectively, in the Fortran variables. In the case of biaxial loading, this subroutine is bypassed as there are no free edges.

**Part 1.—Decay length or boundary layer width computations.** The interlaminar stresses near the free edge are assumed to decay exponentially. The decay length is calculated with the aid of the following equations:

$$\{\ell_b\} = \frac{-\alpha_{if}}{\lambda} \left( \frac{\ell_f}{\ell_c} \right)$$

where

$$\alpha = \bar{G}^{-1} (0.001)$$

and

$$\{\lambda\} = \left\{ \frac{G_m}{E_{hy}} \left[ \sqrt{\frac{\pi}{4(1-k_y)k_f}} - 1 \right] \right\}^{1/2}$$

The calculations are repeated for each layer. Quantities  $\ell_b$  and  $\lambda_i$  are stored in arrays YPL and PLMDAY. These quantities pertain to the free edge parallel to the load axis  $X$ . The corresponding quantities for the load axis parallel to  $Y$  are stored in arrays XPL and PLMDAX. These are computed by replacing  $E_{hy}$  with  $E_{hx}$  in the preceding equations. For the intraply hybrid composite, the respective arrays for the secondary composite are denoted by YSL, SLMDAY, XSL, and SLMDAX. Note that the letter P is replaced by S. This notation is followed consistently throughout the text. The labeled common block ILAB6 is used to store and pass these data to subroutine EDGSTR.

**Part 2.—Interlaminar stress computations.** In the EDGSTR subroutine, the ply stresses PL(67,I) to PL(69,I) are transformed to the structural coordinate system  $x$ ,  $y$ , and  $z$ . These stresses are stored in the matrix SIGMA (3,I) for each layer. The interlaminar stresses  $\{\sigma_{zx}\}$  are computed with the aid of the following relations:

$$\sigma_{zx}^i = \alpha^2 \left( \frac{\ell_f^i}{\ell_b} \right)^2 \left[ \frac{\sigma_{hy}^i}{2} + \frac{1}{\ell_f^i} \sum_{j=N_f}^{j+1} \sigma_{hy}^j \ell_f^j \right]$$

for  $i = N_{f-1}$  to  $N_f/2 + 1$

$$\sigma_{zz}^{N_f} = \alpha^2 \left( \frac{\ell_f^{N_f}}{L_b^{N_f}} \right)^2 \frac{\sigma_{hy}^{N_f}}{2}$$

The interlaminar shear stresses  $\{\sigma_{zy}\}$  and  $\{\sigma_{zx}\}$  are calculated by

$$\sigma_{zy}^i = \frac{\alpha}{(e\alpha - 1)} \frac{\sum_{j=N_f}^{j+1} \sigma_{hy}^j \ell_f^j}{\ell_b} \quad \text{for } i = N_f \text{ to } \frac{N_f}{2} + 1$$

and

$$\sigma_{bx}^i = \frac{3 \sum_{j=N_i}^{j+1} \sigma_{by}}{\delta_b} \quad \text{for } i = N_i \text{ to } \frac{N_i}{2} + 1$$

In these equations, the computations are started from the top layer ( $i = N_i$ ). After the midplane is approached ( $i = N_i/2 + 1$ ), the calculations are repeated starting from the bottom layer ( $i = 1$ ) and continued until  $i$  becomes ( $N_i - 1$ ).

The interlaminar stresses are stored in the arrays YSZZP, SZYP, and SZXP for the primary composite and in the arrays YSZZS, SZYS, and SZXS for the secondary composite. They are, however, made dimensionless by dividing by the applied normal stress  $\sigma_{bx}$ .

**Subroutine STRCNF.**—This subroutine calculates the stress concentration factors around a circular hole due to membrane loading. The equations used are taken from reference 19 and are strictly applicable for infinite plates. Three factors are computed in the subroutine and are defined by the following equations:

$$K_{1xx} = \frac{\sigma_{\theta\theta}}{\sigma_{xx\infty}}$$

$$K_{1yy} = \frac{\sigma_{\theta\theta}}{\sigma_{yy\infty}}$$

$$K_{1xy} = \frac{\sigma_{\theta\theta}}{\sigma_{xy\infty}}$$

Quantities  $\sigma_{xx\infty}$ ,  $\sigma_{yy\infty}$ , and  $\sigma_{xy\infty}$  are the applied stresses, and  $\sigma_{\theta\theta}$  is the hoop stress at any angle  $\theta$  from the load axis. The stress concentration factors are stored in the local arrays XK1, XK3, and TEMP. The expressions for  $K_{1xx}$ ,  $K_{1yy}$ , and  $K_{1xy}$  are the following:

$$\begin{aligned} K_{1xx} &= \frac{E_{c1t}}{E_{c1r}} \left\{ -\sqrt{\frac{E_{c1r}}{E_{c1t}}} \cos^2 \theta + \left[ 1 + \sqrt{2 \left( \frac{E_{c1r}}{E_{c1t}} - \nu_{c1r} \right)} + \frac{E_{c1r}}{G_{c1r}} \right] \sin^2 \theta \right\} \\ K_{1yy} &= \frac{E_{c1r}}{E_{c1t}} \left\{ -\sqrt{\frac{E_{c1t}}{E_{c1r}}} \cos^2 \theta + \left[ 1 + \sqrt{2 \left( \frac{E_{c1t}}{E_{c1r}} - \nu_{c1t} \right)} + \frac{E_{c1t}}{G_{c1t}} \right] \sin^2 \theta \right\} \\ K_{1xy} &= \frac{E_{c1t}}{E_{c1r}} \left\{ 1 + \sqrt{\frac{E_{c1r}}{E_{c1t}}} + \left[ \sqrt{2 \left( \frac{E_{c1r}}{E_{c1t}} - \nu_{c1r} \right)} + \frac{E_{c1r}}{G_{c1r}} \right] \right. \\ &\quad \left. - \left[ \sqrt{2 \left( \frac{E_{c1r}}{E_{c1t}} - \nu_{c1r} \right)} + \frac{E_{c1r}}{G_{c1r}} \right] \sin 2\theta \right\} \end{aligned}$$

In the preceding expressions,  $E_{c1t}$  and  $E_{c1r}$  are the composite moduli in the tangential and radial directions at angle  $\theta$ . Angle  $\theta$  is measured from the x-axis for  $K_{1xx}$  and  $K_{1xy}$  and from the y-axis for  $K_{1yy}$ . The program rearranges the computed  $K_{1yy}$  values so that they correspond to the same location as those of  $K_{1xx}$  and  $K_{1xy}$ .

**Subroutine NUDIFS.**—In this subroutine, the Poisson's ratio differences between the adjacent plies and the composite are computed around a circular hole at 5° intervals. The products of the differences and the corresponding stress concentration factors are computed next. These products are expected to provide insight into the probable delamination locations. It is assumed that onset of

delamination is likely to occur at the locations for which the product of Poisson's ratio mismatch with the corresponding stress concentration factor is a maximum. Accordingly, these products are computed at 5° intervals and the maxima are calculated. Two sets of tables are the output from this subroutine. The first table comes out optionally if the boolean NONUDF is set to FALSE. It contains all the details of the computations. The second table consists of the summary of results, with notes on the maxima and the locations. The following are the programmed equations:

At any angle  $\theta$  the Poisson's ratio is computed by

$$\nu_{cr1} = E_{cr1} \left[ \frac{\nu_{cxy}}{E_{cxx}} - \left( \frac{1 + 2\nu_{cxy}}{E_{cxx}} + \frac{1}{E_{cyy}} - \frac{1}{G_{cxy}} \right) \cos^2 \theta \sin^2 \theta \right]$$

The ply Poisson's ratio is given by

$$\{\nu_{tri}\} = E_{tri} \left[ \frac{\nu_{n2}}{E_{n1}} - \left( \frac{1 + 2\nu_{n2}}{E_{n1}} + \frac{1}{E_{n2}} - \frac{1}{G_{n2}} \right) \cos^2 \theta \sin^2 \theta \right]$$

The difference in Poisson's ratio between the  $i$ th and  $(i+1)$ th plies is given by  $(\nu_{tri}^{i+1} - \nu_{tri}^i)$ , and the difference with respect to the composite is given by  $(\nu_{tri}^i - \nu_{cr1})$ . These are stored in the arrays A2 and A3, respectively. The products of  $K_{1xx}$ ,  $K_{1yy}$ , and  $K_{1xy}$  with A3 are computed next and are stored in the arrays A5, A6, and A7, respectively. The maxima and their location in each of the four quadrants (0-90, 90-180, 180-270, and 270-0) are computed by calling the subroutine AMAXF for the three arrays A5, A6, and A7. The values of stress concentration factors are passed through the labeled common block ILAB8 from the subroutine STRCNF.

**Subroutine MSCBFL (AINF).**—A complete laminate failure stress analysis, based on first-ply failure and the maximum strength criteria, is performed in this subroutine. The inputs to this routine are the ply allowables  $S_{n1C}$ ,  $S_{n1T}$ ,  $S_{n2C}$ ,  $S_{n2T}$ , and  $S_{n2S}$  and the ply influence coefficient matrix AINF. The ply stress allowables are generated by the INHYD routines and are stored in the ply properties array PL. These are accessed through the labeled common block ILAB2. The ply stress influence coefficients are generated by COMSA and the main program and are passed to the present routine by the subroutine argument.

The failure stress for a particular ply due to a specific loading is given by the ratio of the allowable strength to the ply stress influence coefficient. For example, the failure stress due to a tensile load is given by

$$S_c^i = \frac{S_{n1T}}{\text{Fact1}^i}$$

where  $\text{Fact1}^i$  is the stress influence coefficient for  $i$ th ply due to unit tensile loading,  $S_{n1T}$  is the strength allowable for  $i$ th ply in longitudinal tension, and  $S_c^i$  is the failure stress for the  $i$ th ply due to a tensile loading. The failure stresses are stored in the matrix FAILD. In the case of temperature/moisture presence, the allowable strengths are updated to take into account temperature or moisture stresses; the failure stresses are computed with and without the effects of temperature- and moisture-induced stresses for comparison. The program considers primarily five different loadings, longitudinal compression and tension, transverse compression, and tension and inplane shear.

After the failure load computations for each ply are determined, the active failure mode and the corresponding failure strength for each type of loading are determined by calling the subroutine AMINF. This subroutine returns the value of the minimum failure load, the ply number, and the failure mode as output. The output from this subroutine is printed under the heading LAMINATE FAILURE STRESS ANALYSIS.

**Subroutine MCRSTR.**—This subroutine generates the microstresses in the ply constituents due to the inplane loading. These are stored in the ply microproperty arrays PLMP and PLMS for the

primary and the secondary composites. The ply constituent properties and the applied loads are inputs to this subroutine. They are accessed with the aid of the common blocks PBANK, MFBANK, ILAB2, ILAB5, and ILAB9. The PLMP and PLMS each contain 41 entries which are explained in the following list:

<i>Code name</i>	<i>Algebraic notation</i>	<i>Fortran variable</i>
PLM(1,I)	$\sigma_{m11L}$	SM1L
PLM(2,I)	$\sigma_{m11T}$	SM1T
PLM(3,I)	$\sigma_{f11L}$	SF1L
PLM(4,I)	$\sigma_{f11T}$	SF1T
PLM(5,I)	$\sigma_{m22L}^{(A)}$	SM2AL
PLM(6,I)	$\sigma_{m22T}^{(A)}$	SM2AT
PLM(7,I)	$\sigma_{m22L}^{(B)}$	SM2BL
PLM(8,I)	$\sigma_{m22T}^{(B)}$	SM2BT
PLM(9,I)	$\sigma_{f22L}^{(B)}$	SF2BL
PLM(10,I)	$\sigma_{f22T}^{(B)}$	SF2BT
PLM(11,I)	$\sigma_{m33L}^{(A)}$	SM3AL
PLM(12,I)	$\sigma_{m33T}^{(A)}$	SM3AT
PLM(13,I)	$\sigma_{m33L}^{(B)}$	SM3BL
PLM(14,I)	$\sigma_{m33T}^{(B)}$	SM3BT
PLM(15,I)	$\sigma_{f33L}^{(B)}$	SF3BL
PLM(16,I)	$\sigma_{f33T}^{(B)}$	SF3BT
PLM(17,I)	$\sigma_{m12}^{(A)}$	SM12A
PLM(18,I)	$\sigma_{m12}^{(B)}$	SM12B
PLM(19,I)	$\sigma_{f12}^{(B)}$	SF12B
PLM(20,I)	$\sigma_{m13}^{(A)}$	SM13A
PLM(21,I)	$\sigma_{m13}^{(B)}$	SM13B
PLM(22,I)	$\sigma_{f13}^{(B)}$	SF13B
PLM(23,I)	$\sigma_{m23}^{(A)}$	SM23A
PLM(24,I)	$\sigma_{m23}^{(B)}$	SM23B

PLM(25,I)	$\sigma_{f23}^{(B)}$		SF23B
PLM(26,I)	$\sigma_{m11}$	Microstresses due to temperature gradient $\Delta T$	SM11DT
PLM(27,I)	$\sigma_{f11}$		SF11DT
PLM(28,I)	$\sigma_{m22}^{(A)}$		SM2ADT
PLM(29,I)	$\sigma_{m22}^{(B)}$		SM2BDT
PLM(30,I)	$\sigma_{f22}^{(B)}$		SF2BDT
PLM(31,I)	$\sigma_{m33}^{(A)}$		SM3ADT
PLM(32,I)	$\sigma_{m33}^{(B)}$		SM3BDT
PLM(33,I)	$\sigma_{f33}^{(B)}$		SF3BDT
PLM(34,I)	$\sigma_{m11}$	Microstresses due to moisture $M_f$	SM11DM
PLM(35,I)	$\sigma_{f11}$		SF11DM
PLM(36,I)	$\sigma_{m22}^{(A)}$		SM2ADM
PLM(37,I)	$\sigma_{m22}^{(B)}$		SM2BDM
PLM(38,I)	$\sigma_{f22}^{(B)}$		SF2BDM
PLM(39,I)	$\sigma_{m33}^{(A)}$		SM3ADM
PLM(40,I)	$\sigma_{m33}^{(B)}$		SM3BDM
PLM(41,I)	$\sigma_{f33}^{(B)}$		SF3BDM

In this list, entries 26 to 41 are suppressed automatically if the temperature gradients and moisture contents are not present. The superscripts A and B refer to two regions as described in figure 4.

The microstresses are calculated with the aid of the following equations: (For notation and sign conventions, see figs. 4 and 6.)

Ply microstresses due to a longitudinal stress  $\sigma_{n1}$  are given by

$$\sigma_{m11} = (E_m/E_{n1})\sigma_{n1}$$

$$\sigma_{n1} = (E_{n1}/E_{n1})\sigma_{n1}$$

$$\sigma_{m22}^{(A)} = (\nu_m - \nu_{n2})(E_m/E_{n1})\sigma_{n1}$$

$$\sigma_{m22}^{(B)} = \sigma_{f22}^{(B)} = - \frac{1 - \sqrt{k_f}}{\sqrt{k_f}} \sigma_{m22}^{(A)}$$

$$\sigma_{m33}^{(A)} = \sigma_{m22}^{(A)}$$



$$\sigma_{m33}^{(B)} = \sigma_{m22}^{(B)}$$

$$\sigma_{f33}^{(B)} = \sigma_{f22}^{(B)}$$

Ply microstresses due to a transverse stress  $\sigma_{22}$  are given by

$$\sigma_{m11} = \left( \nu_m - \frac{\nu_{f2} E_m}{E_{f1}} \right) \sigma_{22}$$

$$\sigma_{f11} = \left( \nu_{f2} - \nu_{f2} \frac{E_{f1}}{E_{f1}} \right) \sigma_{22}$$

$$\sigma_{m22}^{(A)} = (E_m/E_2) \sigma_{22}$$

$$\sigma_{m22}^{(B)} = (E_{f2}/E_2) \sigma_{22}$$

$$\sigma_{f22}^{(B)} = (E_{f2}/E_2) \sigma_{22}$$

where  $E_2$  is given by

$$E_2 = (1 - \sqrt{k_f}) E_m + \frac{\sqrt{k_f} E_m}{1 - \sqrt{k_f} \left( 1 - \frac{E_m}{E_{f2}} \right)}$$

$$\sigma_{m33}^{(A)} = (\nu_m/\nu_{f3})(E_m/E_{f2}) \sigma_{22}$$

$$\sigma_{m33}^{(B)} = - \frac{1 - \sqrt{k_f}}{\sqrt{k_f}} \sigma_{m33}^{(A)}$$

$$\sigma_{f33}^{(B)} = \sigma_{m33}^{(B)}$$

Ply microstresses due to inplane shear stress ( $\sigma_{12}$ ) are given by

$$\sigma_{m12}^{(A)} = (G_m/G_{12}) \sigma_{12}$$

$$\sigma_{m12}^{(B)} = (G_{f2}/G_{12}) \sigma_{12}$$

$$\sigma_{f12}^{(B)} = (G_{f2}/G_{12}) \sigma_{12}$$

where  $G_{12}$  is given by

$$G_{12} = \left(1 - \sqrt{k_f}\right) G_m + \frac{\sqrt{k_f} G_m}{1 - \sqrt{k_f} \left(1 - \frac{G_m}{G_{23}}\right)}$$

$$\sigma_{m13}^{(A)} = (G_m / G_{12}) \sigma_{n3}$$

$$\sigma_{m13}^{(B)} = (G_{n2} / G_{12}) \sigma_{n3}$$

$$\sigma_{f13}^{(B)} = \sigma_{m13}^{(B)}$$

Ply microstresses due to through-the-thickness shear stress  $\sigma_{n3}$  are given by

$$\sigma_{m23}^{(A)} = (G_m / G_{23}) \sigma_{n3}$$

$$\sigma_{m23}^{(B)} = (G_{23} / G_{23}) \sigma_{n3}$$

where  $G_{23}$  is given by

$$G_{23} = \frac{G_m}{1 - \sqrt{k_f} \left(1 - \frac{G_m}{G_{23}}\right)}$$

$$\sigma_{f23}^{(B)} = \sigma_{m23}^{(B)}$$

Ply microstresses due to temperature gradient  $\Delta T_f$  are given by

$$\sigma_{m11} = (\alpha_{n1} - \alpha_m) \Delta T_f E_m$$

$$\sigma_{f11} = (\alpha_{n1} - \alpha_{f1}) \Delta T_f E_{f1}$$

$$\sigma_{m22}^{(A)} = (\alpha_{n2} - \alpha_m) \Delta T_f E_m$$

$$\sigma_{m22}^{(B)} = \sigma_{f22}^{(B)} = - \frac{1 - \sqrt{k_f}}{\sqrt{k_f}} \sigma_{m22}^{(A)}$$

$$\sigma_{m33}^{(A)} = \sigma_{m22}^{(A)}$$

$$\sigma_{m33}^{(B)} = \sigma_{m22}^{(B)}$$

PINF(10,K,NLD)	$\sigma_{m33}^{(A)}$	SM33A
PINF(11,K,NLD)	$\sigma_{m33}^{(B)}$	SM33B
PINF(12,K,NLD)	$\sigma_{f11}$	SF11
PINF(13,K,NLD)	$\sigma_{f22}^{(B)}$	SF22B
PINF(14,K,NLD)	$\sigma_{f33}^{(B)}$	SF33B
PINF(15,K,NLD)	$\sigma_{f12}$	SF12
PINF(16,K,NLD)	$\sigma_{f13}$	SF13
PINF(17,K,NLD)	$\sigma_{f23}^{(B)}$	SF23B

The dimension K varies from 1 to NMS, where NMS is the number of material systems. NLD varies from 1 to 7. The expression NLD = 1 to 5 refers to unit applied stresses in 11, 22, 12, 13, and 23, respectively. The expression NLD = 6 corresponds to unit temperature loading, and the expression NLD = 7 corresponds to unit moisture loading.

The microstress influence coefficients are computed for secondary composites and optionally computed for intraply hybrid composites. These are stored in the matrix SINF.

**Subroutines AMAXF, AMINF, LOGO, and LOGO2.**—These subroutines perform several auxiliary duties. AMAXF finds the maximum value of a one-dimensional array and its location. AMINF finds the minimum value of a one-dimensional array and its location. These two subroutines are utilized by MSCBFL and NUDIFS in conjunction with searching for failure loads and the probable locations of delamination. LOGO is a subroutine to generate the ICAN emblem for the output. The description of the material and the structural coordinate system by appropriate figures is generated by the subroutine LOGO2.

**Subroutine INHYD.**—This subroutine generates the composite ply properties, necessary for the laminate response analysis. The inputs to this routine are the constituent properties which are supplied in the appropriate format by the subroutines IDGED and BANKRD. INHYD calls the subroutines FIBMT, COMPP, and HTM to perform the micromechanics analysis, including the analysis of hygrothermal effects. The ply properties are stored in the array PROPS, which is accessed by the main program through the labeled common block PBANK. INHYD is called once for each different material system by the main program. The outputs of INHYD show the properties of the fiber, matrix, and composite.

The fiber and matrix properties for the primary composite are read in from the input provided by IDGER. These are stored in arrays PF and PM. Similarly, the arrays SF and SM are used to store the properties of secondary composite constituents if the composite is of the hybrid type. The program then checks for temperature and moisture. The properties of the matrix are updated for the presence of temperature and moisture. The following are the equations programmed to account for the hygrothermal property degradation:

The wet glass transition temperature is computed from

$$T_{gwr} = (0.005M_f^2 - .1M_f + 1)T_{gdr}$$

where  $T_{gwr}$  is the wet glass transition temperature,  $T_{gdr}$  is the dry glass transition temperature for the resin matrix, and  $M_f$  is the percentage of moisture by weight.

The reduction factors  $X_{mp}$  and  $X_{lp}$  are computed from

$$X_{mp} = \sqrt{(T_{gwr} - T_u) / (T_{gdr} - T_o)}$$

$$X_{lp} = 1 / X_{mp}$$

where  $T_o$  is the reference temperature (70 °F), and  $T_u$  is the use temperature.

$$\sigma_{f33}^{(B)} = \sigma_{f22}^{(B)}$$

$$\Delta T_f = T_f - T_{cu}$$

Ply microstresses due to moisture  $M_f$  are given by

$$\sigma_{m11} = (\beta_{n1} - \beta_m) M_f E_m$$

$$\sigma_{f11} = \beta_{n1} M_f E_{f11}$$

$$\sigma_{m22}^{(A)} = (\beta_{n2} - \beta_m) M_f E_m$$

$$\sigma_{m22}^{(B)} = - \frac{1 - \sqrt{k_f}}{\sqrt{k_f}} \sigma_{m22}^{(A)}$$

$$\sigma_{m33}^{(A)} = \sigma_{m22}^{(A)}$$

$$\sigma_{m33}^{(B)} = \sigma_{m22}^{(B)}$$

$$\sigma_{f33}^{(A)} = \sigma_{f33}^{(A)}$$

**Subroutine MINCOF.**—This subroutine generates the microstress influence coefficients for each different material system used in the layup. The equations used are similar to those programmed for MCRSTR. However, the influence coefficients are based on the application of unit load in a specific direction or unit temperature difference or unit moisture content. The influence coefficients are stored in the matrix PINF. This matrix has 17 entries. They are described in the following list:

Entry	Algebraic notation	Fortran variable
PINF(1,K,NLD)	$\sigma_{m11}$	SM11
PINF(2,K,NLD)	$\sigma_{m22}^{(A)}$	SM22A
PINF(3,K,NLD)	$\sigma_{m22}^{(B)}$	SM22B
PINF(4,K,NLD)	$\sigma_{m12}^{(A)}$	SM12A
PINF(5,K,NLD)	$\sigma_{m12}^{(B)}$	SM12B
PINF(6,K,NLD)	$\sigma_{m13}^{(A)}$	SM13A
PINF(7,K,NLD)	$\sigma_{m13}^{(B)}$	SM13B
PINF(8,K,NLD)	$\sigma_{m23}^{(A)}$	SM23A
PINF(9,K,NLD)	$\sigma_{m23}^{(B)}$	SM23B

The moduli and strengths of the matrix are multiplied by  $X_{mp}$  to obtain the new properties for the matrix. The density is given by

$$\rho_{mw} = \rho_m + 3\rho_m k_m M / 100$$

The thermal properties, such as heat capacity, thermal expansion coefficient, and thermal conductivity are multiplied by the second factor  $X_{tp}$  to account for the hygrothermal conditioning.

After the property arrays PF, PM, SF, and SM are properly filled, the program chooses either FIBMT or HTM subroutines to perform micromechanics. The subroutine HTM is chosen if temperature/moisture effects are to be taken into consideration. Otherwise, FIBMT is chosen for dry room temperature property computations. The outputs from these routines are primary and secondary composite ply properties. They are stored in the arrays P and S, respectively. These properties are made common to subroutine COMPP through the common blocks ILAB1 and ILAB3. The subroutine COMPP is called by INHYD for hybrid composites to compute the hybrid composite ply properties. These properties are stored in the array H. One of the arrays P, S, or H are passed to ICAN via common block PBANK and the array PROPS. For example, if the ply is made of 100 percent primary composite only, the array PROPS is assigned to have the same entries as P, etc.

The subroutine INHYD also calls FLEXX, which performs a flexural strength analysis. However, these are only for additional information and are not used by ICAN at the present time.

**Subroutine FIBMT (C, F, M, VF, VM, VP, KV, IFLAG).**—This subroutine generates properties of a ply by using the constituent properties which are supplied from the subroutine INHYD. The constituent properties are stored in the arrays F and M; F contains the fiber properties, and M contains the matrix properties. The composite properties are stored in the array C, which is returned to INHYD. The theory behind the programmed equations is discussed in reference 13. The following is a description of each entry in the arrays C, F, and M, with the corresponding algebraic notation:

#### Composite Properties Array C(I)

Entry	Description	Notation
C(1)	elastic moduli	$E_{n1}$
C(2)	elastic moduli	$E_{n2}$
C(3)	elastic moduli	$E_{n3}$
C(4)	shear moduli	$G_{n2}$
C(5)	shear moduli	$G_{n3}$
C(6)	shear moduli	$G_{n3}$
C(7)	Poisson's ratio	$\nu_{n2}$
C(8)	Poisson's Ratio	$\nu_{n3}$
C(9)	Poisson's Ratio	$\nu_{n3}$
C(10)	thermal expansion coefficient	$\alpha_{n1}$
C(11)	thermal expansion coefficient	$\alpha_{n2}$
C(12)	thermal expansion coefficient	$\alpha_{n3}$
C(13)	density	$\rho_f$
C(14)	heat capacity	$C_f$
C(15)	heat conductivity	$K_{n1}$
C(16)	heat conductivity	$K_{n2}$
C(17)	heat conductivity	$K_{n3}$
C(18)	strength	$S_{n1T}$
C(19)	strength	$S_{n1C}$
C(20)	strength	$S_{n2T}$
C(21)	strength	$S_{n2C}$
C(22)	strength	$S_{n2}$
C(23)	moisture diffusivity	$D_{n1}$
C(24)	moisture diffusivity	$D_{n2}$
C(25)	moisture diffusivity	$D_{n3}$
C(26)	moisture expansion coefficient	$\beta_{n1}$
C(27)	moisture expansion coefficient	$\beta_{n2}$

C(28)	moisture expansion coefficient	$\beta_{n3}$
C(29)	flexural moduli	$E_{n1}$
C(30)	flexural moduli	$E_{n2}$
C(31)	strengths (flexural)	$S_{n3}$
C(32)	strengths (flexural)	$S_{n1F}$
C(33)	strengths (flexural)	$S_{n2F}$
C(34)	strengths (flexural)	$S_{n2}$
C(35)	ply thickness	$t_f$
C(36)	interply thickness	$\delta_f$
C(37)	interfiber spacing	$\delta_s$

#### Fiber Properties Array

Entry	Description	Notation
F(1)	elastic moduli	$E_{f1}$
F(2)	elastic moduli	$E_{f2}$
F(3)	shear moduli	$G_{f2}$
F(4)	shear moduli	$G_{f2}$
F(5)	Poisson's ratio	$\nu_{f12}$
F(6)	Poisson's ratio	$\nu_{f23}$
F(7)	thermal expansion coefficient	$\alpha_{f1}$
F(8)	thermal expansion coefficient	$\alpha_{f2}$
F(9)	density	$\rho_f$
F(10)	number of fibers per end	$N_f$
F(11)	fiber diameter	$d_f$
F(12)	heat capacity	$C_f$
F(13)	heat conductivity	$K_{f1}$
F(14)	heat conductivity	$K_{f2}$
F(15)	heat conductivity	$K_{f3}$
F(16)	strength	$S_{fT}$
F(17)	strength	$S_{fC}$

#### Matrix Properties Array

Entry	Description	Notation
M(1)	elastic modulus	$E_m$
M(2)	shear modulus	$G_m$
M(3)	Poisson's ratio	$\nu_m$
M(4)	thermal expansion coefficient	$\alpha_m$
M(5)	density	$\rho_m$
M(6)	heat capacity	$C_m$
M(7)	heat conductivity	$K_m$
M(8)	strength	$S_{mT}$
M(9)	strength	$S_{mC}$
M(10)	strength	$S_{mS}$
M(11)	moisture coefficient	$\beta_m$
M(12)	diffusivity	$D_m$

The following are the programmed equations for the entries in array C:

**Normal moduli:**

$$E_{n1} = k_f E_{f1} + k_m E_m$$

$$E_{n2} = \frac{E_m}{1 - \sqrt{k_f}(1 - E_m/E_{f22})}$$

$$E_{n3} = E_{n2}$$

**Shear moduli:**

$$G_{n2} = \frac{G_m}{1 - \sqrt{k_f}\left(1 - \frac{G_m}{G_{f12}}\right)}$$

$$G_{n3} = G_{n2}$$

$$G_{n3} = \frac{G_m}{1 - k_f\left(1 - \frac{G_m}{G_{f23}}\right)}$$

**Poisson's Ratio:**

$$\nu_{n2} = \nu_m + k_f(\nu_{f12} - \nu_m)$$

$$\nu_{n3} = \nu_{n2}$$

$$\nu_{n3} = k_f \nu_{f23} + k_m \left( 2\nu_m - \frac{\nu_{n2}}{E_{n1}} E_{n2} \right)$$

**Coefficients of thermal expansion:**

$$\alpha_{n1} = \frac{\alpha_{f1} + k_m[(\alpha_m E_m/E_{f1}) - \alpha_{f1}]}{1 + k_m\left(\frac{E_m}{E_{f1}} - 1\right)}$$

$$\alpha_{22} = \alpha_m(1 - \sqrt{k_f}) \left[ \frac{1 + k_f \nu_m E_{f1}}{E_{f1} + k_m(E_m - E_{f1})} \right] + \alpha_{f22} k_f$$

$$\alpha_{33} = \alpha_{22}$$

**Density:**

$$\rho_t = \rho_f k_f + \rho_m k_m$$

**Heat capacity:**

$$C_t = \frac{(k_f C_p \rho_f + k_m C_m \rho_m)}{\rho_t}$$

**Heat conductivities:**

$$K_{n1} = k_f K_{f1} + k_m K_m$$

$$K_{22} = (1 - \sqrt{k_f}) K_m + \frac{\sqrt{k_f}}{1 - \sqrt{k_f} \left(1 - \frac{K_m}{K_{f22}}\right)} K_m$$

$$K_{33} = K_{22}$$

In the preceding equations,  $K_m$  should be replaced by

$$K_m - (1 - \sqrt{k_v}) K_m + \frac{K_m \sqrt{k_v}}{1 - \sqrt{k_v} \left(1 - \frac{K_m}{K_v}\right)}$$

if there are voids. The quantity  $K_v$  is the void conductivity.

**Strengths:**

$$S_{n1T} = S_{fT}(k_f + k_m E_m / E_{f1})$$

The longitudinal compressive strength is computed based on three different criteria, rule of mixtures, fiber microbuckling, and delamination. The minimum of the three estimates is returned as  $S_{n1C}$ . The equations for the three cases are



$$S_{11C} \text{ (rule of mixtures)} = S_{fc}(k_f + k_m E_m / E_{f1})$$

$$S_{11C} \text{ (delamination)} = (13S_{n2} + S_{mc})$$

$$S_{11C} \text{ (fiber microbuckling)} = \frac{F_2 G_m}{1 - k_f \left(1 - \frac{G_m}{G_{f2}}\right)}$$

The transverse strengths are calculated from

$$S_{22T} = S_{mT}(\text{FACT}/\text{DENOM})$$

$$S_{22C} = S_{mc}/\text{DENOM}$$

$$S_{n2} = \frac{[(F_1 - 1 + G_m/G_{f2})F_2 G_{n2} S_{ms}]}{G_m F_1} \text{FACT}$$

where  $F_1$  and  $F_2$  are defined by the equations:

$$F_1 = \sqrt{\frac{\pi}{4k_f}}$$

$$F_2 = 1 - \sqrt{\frac{4k_p}{\pi k_m}}$$

The variable DENOM is a Fortran variable given by

$$\text{DENOM} = \left[1 - \sqrt{k_f} \left(1 - \frac{E_m}{E_{f22}}\right)\right] \sqrt{1 + \varphi(\varphi - 1) + \frac{1}{3}(\varphi - 1)^2}$$

where  $\varphi$  is given by

$$\varphi = \frac{F_1 - \frac{E_m}{E_{f22} \left[1 - \sqrt{k_f} \left(1 - \frac{E_m}{E_{f22}}\right)\right]}}{F_1 - 1}$$

The Fortran variable FACT takes the value  $k_m$  if IFLAG is unity. Otherwise FACT takes the value unity. This variable is introduced to correlate the strengths of HMS and Kevlar fiber composites with the experimentally observed values. The main program INHYD checks for these fibers and assigns the appropriate values for IFLAG. IFLAG is set at zero for other fibers.

**Moisture diffusivities:**

$$D_{n1} = k_m D_m$$

$$D_{n2} = (1 - \sqrt{k_f}) D_m$$

$$D_{n3} = D_{n2}$$

**Moisture expansion coefficients:**

$$\beta_{n1} = \beta_m k_m E_m / E_{n1}$$

$$\beta_{n2} = \beta_m (1 - \sqrt{k_f}) (1 + k_f E_m / E_{n1})$$

$$\beta_{n3} = \beta_{n2}$$

**Flexural moduli ( $E_{n1F}$ ,  $E_{n2F}$ ):**

$$E_{n1F} = E_{n1}$$

$$E_{n2F} = E_{n2}$$

**Flexural Strengths:**

$$S_{n3F} = \frac{\left(F_1 - 1 + \frac{G_m}{G_{f23}}\right) F_2 G_{n3} S_{m3}}{G_m F_1}$$

$$S_{n2F} = 1.5 S_{n2}$$

**Ply thickness:** A default value of 0.005 is set for  $t_p$ . This is overridden by the user specified value in the ICAN main program.

**Interply thickness and interfiber spacing:**

$$\delta_t = \left(\sqrt{\frac{\pi}{k_f}} - 2\right) \frac{d_f}{2}$$

$$\delta_f = \delta_t$$

**Subroutine HTM (C, F, M, VF, VM, VV, IFLAG).**—This subroutine generates the hygrothermomechanical properties based on the theory proposed in reference 15. The subroutine is called only if nontrivial entries for the use temperature and the moisture content ( $T_u \neq 70$  °F or nonzero moisture content) are present. The equations programmed are mostly those discussed in the subroutine FIBMT description. Therefore, only the equations which are different are mentioned here.

**Moisture expansion coefficients:**

$$\beta_{n2} = (1 - \sqrt{k_f}) \beta_m \left[ 1 + \frac{\sqrt{k_f}(1 - \sqrt{k_f})E_m}{\sqrt{k_f}E_{n2} + (1 - \sqrt{k_f})E_m} \right]$$

$$\beta_{n3} = \beta_{n2}$$

**Strengths:**

$$S_{n2T} = \left( \frac{S_{mT}}{E_m} \right) \frac{E_{n2} \left( 1 - \sqrt{\frac{4k_p}{\pi k_m}} \right) (1 - \sqrt{k_f})}{1 - (\sqrt{k_f}E_{n2}/E_{f22})} \text{FACT}$$

$$S_{n2C} = \left( \frac{S_{mC}}{E_m} \right) \frac{E_{n2} \left( 1 - \sqrt{\frac{4k_p}{\pi k_m}} \right) (1 - \sqrt{k_f})}{1 - (\sqrt{k_f}E_{n2}/E_{f22})} \text{FACT}$$

$$S_{n12} = \left( \frac{S_{mS}}{G_m} \right) G_{n2} \frac{\left( 1 - \sqrt{\frac{4k_p}{\pi k_m}} \right) (1 - \sqrt{k_f})}{1 - (\sqrt{k_f}G_{n2}/G_{f12})} \text{FACT}$$

$$S_{n3F} = \left( \frac{S_{mS}}{G_m} \right) \frac{G_{n3} \left( 1 - \sqrt{\frac{4k_p}{\pi k_m}} \right) (1 - \sqrt{k_f})}{1 - (\sqrt{k_f}G_{n3}/G_{f23})}$$

$$S_{n2F} = 1.5S_{n2}$$

In the preceding equations, FACT is a Fortran variable which is given by

$$\text{FACT} = \delta_f / \delta_f$$

for Kevlar and HMS fibers. For all other fibers FACT = 1.

**Subroutine FLEXX (C).**—The entries C(32) and C(33) of the ply property array C are generated in this subroutine. They are, respectively, the longitudinal flexural strength and the transverse flexural strength. The longitudinal flexural strength is given by

$$S_{n1F} = \frac{2.5S_{n1T}}{\left(1 + \frac{S_{n1T}}{S_{n1C}}\right)}$$

The transverse flexural strength is given by

$$S_{n2F} = \frac{2.5S_{n2T}}{\left(1 + \frac{S_{n2T}}{S_{n2C}}\right)}$$

**Subroutine COMPP (IPFLAG, ISFLAG).**—This subroutine is called by INHYD to generate the properties of a hybrid ply. The equations are based on the theory proposed in reference 13. The properties are stored in the array H. The entries are, however, the same as those of array C given in the description for subroutine FIBMT. The inputs to this routine are the primary composite properties array P, the secondary composites property array S, and the percentage of the secondary composite  $k_{sc}$ . The equations are the following:

**Elastic normal moduli:**

$$E_{n1}(H) = E_{n1}(P) + [E_{n1}(S) - E_{n1}(P)]k_{sc}$$

$$E_{n2}(H) = \frac{E_{n2}(P)}{1 + k_{sc}[E_{n2}(P)/E_{n2}(S) - 1]}$$

$$E_{n3}(H) = E_{n3}(P) + [E_{n3}(S) - E_{n3}(P)]k_{sc}$$

**Shear moduli:**

$$G_{n3}(H) = \frac{G_{n3}(P)}{1 - k_{sc}\left(1 - \frac{G_{n3}(P)}{G_{n3}(S)}\right)}$$

$$G_{n2}(H) = \frac{G_{n2}(P)}{1 - k_{sc}\left(1 - \frac{G_{n2}(P)}{G_{n2}(S)}\right)}$$

$$G_{n3}(H) = G_{n3}(P) + k_{sc}[G_{n3}(S) - G_{n3}(P)]$$

Poisson's ratios:

$$\nu_{n2}(H) = \nu_{n2}(P) + k_{sc}[\nu_{n2}(S) - \nu_{n2}(P)]$$

$$\nu_{n3}(H) = \nu_{n2}(P) + \frac{k_{sc}[\nu_{n2}(P) - \nu_{n2}(S)]}{(1 - k_{sc})[E_{n3}(P)/E_{n3}(S)] - k_{sc}}$$

$$\nu_{n3}(H) = \nu_{n3}(P) + k_{sc}[\nu_{n3}(S) - \nu_{n3}(P)]$$

Coefficients of thermal expansion:

$$\alpha_{n1}(H) = \frac{\alpha_{n1}(P) + k_{sc}[\{\alpha_{n1}(S)E_{n1}(S)/E_{n1}(P)\} - \alpha_{n1}(P)]}{1 + k_{sc}\left(\frac{E_{n1}(S)}{E_{n1}(P)} - 1\right)}$$

$$\alpha_{n3}(H) = \frac{1}{E_{n3}(H)} \left\{ -\nu_{n3}(H)E_{n3}(H)\alpha_{n1}(H) + (1 - k_{sc})E_{n3}(P) \left[ \alpha_{n2}(P) + \nu_{n3}(P)\alpha_{n1}(P) \right] \right. \\ \left. + k_{sc}E_{n3}(S) \left[ \alpha_{n2}(S) + \nu_{n3}(S)\alpha_{n1}(S) \right] \right\}$$

$$\alpha_{n2}(H) = (1 - k_{sc}) \left\{ \alpha_{n2}(S) \left[ 1 + \nu_{n3}(P) \right] + \nu_{n2}(P)\alpha_{n1} \right\} \\ + k_{sc} \left\{ \alpha_{n2}(S) \left[ 1 + \nu_{n3}(S) \right] + \nu_{n2}(S)\alpha_{n1}(S) \right\} \\ - \nu_{n2}(H)\alpha_{n1}(H) - \nu_{n3}(H)\alpha_{n3}(H)$$

Density:

$$\rho_f(H) = (1 - k_{sc})\rho_f(P) + k_{sc}\rho_f(S)$$

Heat capacity:

$$C_f(H) = \left\{ (1 - k_{sc})[C_f(P)k_f(P)\rho_f(P) + C_m(P)k_m(P)\rho_m(P)] \right. \\ \left. + k_{sc}[C_f(S)k_f(S)\rho_f(S) + C_m(S)k_m(S)\rho_m(S)] \right. \\ \left. + [k_f(P)k_v(P) + k_{sc}k_v(S)]M\rho_{msf}C_{msf} \right\} / \rho_f(H)$$

where  $\rho_{msl}$  and  $C_{msl}$  are the moisture density and heat capacity, respectively.

**Heat conductivities:**

$$K_{n1}(H) = (1 - k_{sc})[k_f(P)K_{n1}(P) + k_m(P)K_m(P)] + k_{sc}[k_f(S)K_{n1}(S) + k_m(S)K_m(S)]$$

$$K_{n2}(P) = \frac{(1 - \sqrt{k_f(P)})K_m(P) + \sqrt{k_f(P)}K_m(P)}{1 - \sqrt{k_f(P)}[1 - K_m(P)/K_{n2}(P)]}$$

$$K_{n2}(S) = \frac{(1 - \sqrt{k_f(S)})K_m(S) + \sqrt{k_f(S)}K_m(S)}{1 - \sqrt{k_f(S)}[1 - K_m(S)/K_{n2}(S)]}$$

$$K_{n2}(H) = \frac{K_{n2}(P)}{1 - k_{sc}[1 - K_{n2}(S)/K_{n2}(P)]}$$

$$K_{n3}(H) = K_{n2}(H)$$

The void conductivity  $K_v$  with moisture content  $M$  is given by  $K_v = MK_{mss}$ . If there are voids in the primary composite,  $K_m(P)$  in the preceding equations for heat conductivities is replaced by

$$K_m(P) = [1 - \sqrt{k_v(P)}]K_m(P) + \frac{\sqrt{k_v(P)}K_m(P)}{[1 - \sqrt{k_v(P)}][1 - K_m(P)/K_v]}$$

Similarly, for the secondary composite,  $K_m(S)$  is replaced by

$$K_m(S) = [1 - \sqrt{k_v(S)}]K_m(S) + \frac{\sqrt{k_v(S)}K_m(S)}{[1 - \sqrt{k_v(S)}][1 - K_m(S)/K_v]}$$

**Strengths.**—The longitudinal strengths are based on the rule of mixtures:

$$S_{n1T}(H) = S_{n1T}(P)(1 - k_{sc}) + S_{n1T}(S)k_{sc}$$

$$S_{n1C}(H) = S_{n1C}(P)(1 - k_{sc}) + S_{n1C}(S)k_{sc}$$

The following are a few intermediate variables defined for convenience in the evaluation of transverse strengths:

$$Q_p = 1 - 2\sqrt{k_v(P)/\pi} [1 - 2\sqrt{k_f(P)/\pi}]$$

$$Q_s = 1 - 2\sqrt{k_v(S)/\pi} [1 - 2\sqrt{k_f(S)/\pi}]$$

$$S_{mC} = \min[S_{mC}(P) \text{ and } S_{mC}(S)]$$

$$S_{mT} = \min[S_{mT}(P) \text{ and } S_{mT}(S)]$$

$$S_{mS} = \min[S_{mS}(P) \text{ and } S_{mS}(S)]$$

$$\text{FACT 1} = k_m(P) \quad (\text{for HMS and Kevlar fibers})$$

$$\text{FACT 2} = k_m(S) \quad (\text{for HMS and Kevlar fibers})$$

$$\text{FACT 1} = \text{FACT 2} = 1 \quad (\text{for all other fibers})$$

$$S_{22}(P) = \frac{(1 - k_{sc})Q_p/E_{p22}(P) \left[ 1 - \sqrt{\frac{k_f(P)}{\pi}} \left( 1 - \frac{E_m(P)}{E_{f22}(P)} \right) \right]}{1 - \sqrt{k_f(P)} \left( 1 - \frac{E_m(P)}{E_{f22}(P)} \right)} S_m(P)$$

$$S_{22}(S) = \frac{\frac{k_{sc}Q_s}{E_{22}(S)} \left[ 1 - \sqrt{\frac{k_f(S)}{\pi}} \left( 1 - \frac{E_m(S)}{E_{f22}(S)} \right) \right]}{1 - \sqrt{k_f(S)} \left( 1 - \frac{E_m(S)}{E_{f22}(S)} \right)} S_m(S)$$

$$\varphi_p = \frac{\sqrt{\frac{\pi}{4k_f(P)}} - \frac{E_m(P)}{E_{f22}(P) \left[ 1 - \sqrt{k_f(P)} \left( 1 - \frac{E_m(P)}{E_{f22}(P)} \right) \right]}}{\sqrt{\frac{\pi}{4k_f(P)}} - 1}$$

$$\varphi_s = \frac{\sqrt{\frac{\pi}{4k_f(S)}} - \frac{E_m(S)}{E_{f22}(S) \left[ 1 - \sqrt{k_f(S)} \left( 1 - \frac{E_m(S)}{E_{f22}(S)} \right) \right]}}{\sqrt{\frac{\pi}{4k_f(S)}} - 1}$$

$$\text{DENOMP} = 1 - \sqrt{k_f(P)} \left( 1 - \frac{E_m(P)}{E_{f22}(P)} \right) \sqrt{1 + \varphi_p(\varphi_p - 1) + \frac{1}{3}(\varphi_p - 1)^2}$$

$$\text{DENOMS} = 1 - \sqrt{k_f(S)} \left( 1 - \frac{E_m(S)}{E_{f22}(S)} \right) \sqrt{1 + \varphi_s(\varphi_s - 1) + \frac{1}{3}(\varphi_s - 1)^2}$$

The transverse and the shear strengths of hybrid composites are given by

$$S_{\sigma 2T}(H) = E_{\sigma 2}(H) \left[ \frac{(1 - k_{sc}) \text{FACT1}}{E_{\sigma 2}(P) \text{DENOMP}} + \frac{k_{sc} \text{FACT2}}{E_{\sigma 2}(S) \text{DENOMS}} \right] S_{mT}$$

$$S_{\sigma 2C}(H) = E_{\sigma 2}(H) \left[ \frac{(1 - k_{sc})}{E_{\sigma 2}(P) \text{DENOMP}} + \frac{k_{sc}}{E_{\sigma 2}(S) \text{DENOMS}} \right] S_{mC}$$

$$S_{\sigma 2}(H) = \frac{2G_{\sigma 2}}{\tau} \left\{ \frac{\left( \frac{(1 - k_{sc}) Q_p}{G_{\sigma 2}(P)} \text{FACT1} \left[ 1 - \sqrt{\frac{k_f(P)}{\tau}} \left( 1 - \frac{G_m(P)}{G_{\sigma 2}(P)} \right) \right] \right)}{1 - \sqrt{k_f(P)} \left( 1 - \frac{G_m(P)}{G_{\sigma 2}(P)} \right)} + \frac{\left( \frac{k_{sc} Q_s}{G_{\sigma 2}(S)} \text{FACT2} \left[ 1 - \sqrt{\frac{k_f(S)}{\tau}} \left( 1 - \frac{G_m(S)}{G_{\sigma 2}(S)} \right) \right] \right)}{1 - \sqrt{k_f(S)} \left( 1 - \frac{G_m(S)}{G_{\sigma 2}(S)} \right)} \right\} S_{mS}$$

Moisture diffusivity:

$$D_f(P) = \frac{1 - \sqrt{k_f(P)} + k_f(P)}{[1 - \sqrt{k_f(P)}] D_f(P)}$$

$$D_f(S) = \frac{1 - \sqrt{k_f(S)} + k_f(S)}{[1 - \sqrt{k_f(S)}] D_f(S)}$$

$$D_{\sigma 1}(H) = (1 - k_{sc}) k_m(P) D_f(P) + k_{sc} k_m(S) D_f(S)$$

$$D_{\sigma 2}(H) = (1 - k_{sc}) [1 - 2\sqrt{k_f(P)}] D_f(P) + k_{sc} [1 - \sqrt{k_f(S)}] D_f(S)$$

$$D_{\sigma 3}(H) = D_{\sigma 2}(H)$$

Moisture expansion coefficients:

$$\beta_{\sigma 1}(P) = \frac{k_m(P) \beta_m(P) E_m(P)}{k_f(P) E_{f11}(P) + k_m(P) E_m(P)}$$

$$\beta_{\sigma 1}(S) = \frac{k_m(S) \beta_m(S) E_m(S)}{k_f(S) E_{f11}(S) + k_m(S) E_m(S)}$$

$$\beta_{\sigma 2}(P) = \beta_m(P) [1 - \sqrt{k_f(P)}] \left\{ 1 + \frac{k_f(P) k_m(P) E_{f11}(P)}{E_{f11}(P) + k_m(P) [E_m(P) - E_{f11}(P)]} \right\}$$



$$\beta_{22}(S) = \beta_m(S) \left[ 1 - \sqrt{k_f(S)} \right] \left\{ 1 + \frac{k_f(S) k_m(S) E_{n1}(S)}{E_{n1}(S) + k_m(S) [E_m(S) - E_{n1}(S)]} \right\}$$

$$\beta_{n1}(H) = \frac{[(1 - k_{sc}) k_m(P) \beta_m(P) E_m(P) + k_{sc} k_m(S) \beta_m(S) E_m(S)]}{E_{n1}(H)}$$

$$\beta_{n3}(H) = \left\{ -\nu_{n3}(H) E_{n3}(H) \beta_{n1}(H) + (1 - k_{sc}) E_{n3}(P) [\beta_{22}(P) + \nu_{n3}(P) \beta_{n1}(P)] + k_{sc} E_{n3}(S) [\beta_{22}(S) + \nu_{n3}(S) \beta_{n1}(S)] \right\} / E_{n3}(H)$$

$$\nu_{n2}(P) = \nu_m(P) + k_f(P) [\nu_{n2}(P) - \nu_m(P)]$$

$$\nu_{n2}(S) = \nu_m(S) + k_f(S) [\nu_{n2}(S) - \nu_m(S)]$$

$$\nu_{n3}(P) = \nu_m(P) + k_f(P) [\nu_{n3}(P) - \nu_m(P)]$$

$$\nu_{n3}(S) = \nu_m(S) + k_f(S) [\nu_{n3}(S) - \nu_m(S)]$$

$$\beta_{22}(H) = (1 - k_{sc}) [\beta_{22}(P) (1 + \nu_{n2}(P)) + \nu_{n2}(P) \beta_{n1}(P)] + k_{sc} [\beta_{22}(S) (1 + \nu_{n2}(S)) + \nu_{n2}(S) \beta_{n1}(S)] - \nu_{n2}(H) \beta_{n1}(H) - \nu_{n3}(H) \beta_{n3}(H)$$

**Flexural moduli:**

$$E_{n1F}(H) = E_{n1}(H)$$

$$E_{n2F}(H) = E_{n2}(H)$$

**Flexural strengths:**

$$S_{n3F}(H) = \frac{2G_{n3}(H)}{\pi} \left\{ \frac{(1 - k_{sc}) Q_p}{G_{n3}(P)} \left[ 1 - \sqrt{\frac{k_f(P)}{\pi}} \left( 1 - \frac{G_m(P)}{G_{n3}(P)} \right) \right] \right. \\ \left. + \frac{k_{sc} Q_s}{G_{n3}(S)} \frac{\left[ 1 - \sqrt{\frac{k_f(S)}{\pi}} \left( 1 - \frac{G_m(S)}{G_{n3}(S)} \right) \right]}{1 - \sqrt{k_f(S)} \left( 1 - \frac{G_m(S)}{G_{n3}(S)} \right)} \right\} S_m(S)$$

$$S_{n2SB}(H) = 1.5 S_{n2}(H)$$

Fiber volume ratio:

$$k_f(H) = k_f(P) + k_{sc}[k_f(S) - k_f(P)]$$

**Subroutines BANKRD and IDGER.**—These two subroutines do preprocessing to generate compatible input data to the subroutine INHYD. The subroutine BANKRD is called first by the ICAN main program. The input to this routine is primarily the data supplied on the material card MATCRD by the user. These cards indicate the coded names for the fiber and matrix, the volume ratios of primary and secondary composites, and their respective fiber, and the matrix and void volume ratios. The subroutine BANKRD has its own data base containing the properties of fibers and matrices of commonly used materials. This data base is assigned to input unit 8. It is named FBMTDATA.BANK. The output of BANKRD are the arrays PFP, PFS, PMP, and PMS. The entries in PFP and PFS are the fiber properties of primary and secondary composites. The entries in PMP and PMS are the matrix properties of primary and secondary composites. These arrays are made common to the main program and the subroutine IDGER through the named common block MFBANK. The entries of PF and PM arrays are explained in the following list:

#### Fiber Property Arrays PFP and PFS

Entry	Description	Notation
1	not used	----
2	fiber density	$\rho_f$
3	normal moduli	$E_{f11}$
4	normal moduli	$E_{f22}$
5	Poisson's ratio	$\nu_{f12}$
6	Poisson's ratio	$\nu_{f23}$
7	shear moduli	$G_{f12}$
8	shear moduli	$G_{f23}$
9	thermal expansion coefficient	$\alpha_{f11}$
10	thermal expansion coefficient	$\alpha_{f22}$
11	heat conductivity	$K_{f11}$
12	heat conductivity	$K_{f22}$
13	heat capacity	$C_f$
14	strengths	$S_{fT}$
15	strengths	$S_{fC}$
16	not used	----
17	not used	----
18	not used	----
19	not used	----
20	number of fibers per end	$N_f$
21	fiber diameters	$d_f$

## Matrix Property Arrays PMP and PMS

Entry	Description	Notation
1	not used	---
2	density	$\rho_m$
3	normal modulus	$E_m$
4	Poisson's ratio	$\nu_m$
5	coefficient of thermal expansion	$\alpha_m$
6	heat conductivity	$K_m$
7	heat capacity	$C_m$
8	tensile strength	$S_{mT}$
9	compressive strength	$S_{mC}$
10	shear strength	$S_{mS}$
11	allowable tensile strain	$\epsilon_{mT}$
12	allowable compressive strain	$\epsilon_{mC}$
13	allowable shear strain	$\epsilon_{mS}$
14	allowable torsional strain	$\epsilon_{mTOR}$
15	void conductivity	$K_v$
16	glass transition temperature	$T_{gdr}$

The coded names for the fiber and matrix are stored in the matrix CODES by the main program. The entries in CODES are explained as follows:

CODES(1,1,1)    coded name of primary fiber  
 CODES(1,2,1)    coded name of primary matrix  
 CODES(2,1,1)    coded name of secondary fiber  
 CODES(2,2,1)    coded name of secondary matrix

The subroutine IDGER takes the information generated by BANKRD and arranges it in a proper format for the subroutine INHYD. These data are transferred to input unit 7 prior to calling INHYD. These data are purged at the end of the program execution.

### Data Base FBMTDATA.BANK.

The constituent properties data base is a unique feature of the computer code ICAN. Its primary aim is to reduce the burden on the user by preparing properly formatted data for the program. The user only needs to specify the coded names for the fiber and matrix. The format of the data is explained in this section so as to enable the user to introduce new contents or to modify existing entries as appropriate to his/her needs. Data for four fibers and three matrices are provided in the present package.

The fiber properties are arranged in five physical cards of 80 column length. The first card contains a four-character code name of a fiber in format A4. The second to the fifth cards start with a two-letter mnemonic to indicate the type of properties that follow. The format on any of these cards is A4, 7E10.3, except for the second card. The second card is in the format A3, I6, 7E10.3. The mnemonics FP, FE, FT, and FS stand for fiber physical, elastic, thermal, and strength-related properties, respectively. The entries on these cards are explained as follows:

card 1 four character coded name for fiber  
 card 2 FP;  $N_f$ ,  $d_f$ ,  $\rho_f$   
 card 3 FE;  $E_{f11}$ ,  $E_{f22}$ ,  $\gamma_{f12}$ ,  $\gamma_{f23}$ ,  $G_{f12}$ ,  $G_{f23}$   
 card 4 FT;  $\alpha_{f11}$ ,  $\alpha_{f22}$ ,  $K_{f11}$ ,  $K_{f22}$ ,  $C_f$   
 card 5 FS;  $S_{fT}$ ,  $S_{fC}$  (The remaining entries are open for future modifications.)

The matrix properties are arranged next after the line OVER END OF FIBER PROPERTIES. The properties have essentially the same format as those for fiber property cards. There are, however, six physical cards for each matrix material. The mnemonics used are MP, ME, MT, MS, and MV. They stand for matrix physical, elastic, thermal, strength-related, and miscellaneous properties, respectively. The format for the first card is A4, and the format for the rest of the cards is A3, 7E10.3. The entries in each card are as follows:

card 1 four character coded name for matrix  
 card 2 MP;  $\rho_m$   
 card 3 ME;  $E_m$ ,  $\nu_m$ ,  $\alpha_m$   
 card 4 MT;  $K_m$ ,  $C_m$   
 card 5 MS;  $S_{mT}$ ,  $S_{mC}$ ,  $S_{mS}$ ,  $\epsilon_{mT}$ ,  $\epsilon_{mC}$ ,  $\epsilon_{mS}$ ,  $\epsilon_{mTOR}$   
 card 6 MV;  $K_m$ ,  $T_{gdr}$

The data base presently contains properties for T-300 (T300), AS graphite (AS-), S-Glass (SGLA), and HMS (HMSF) fibers. The available matrix materials are high-modulus, high-strength (HMHS), intermediate-modulus, high-strength (IMHS), and intermediate-modulus, low-strength (IMLS) matrices, which are epoxy-type resins. The complete list of properties is shown in appendix C.

Lewis Research Center  
 National Aeronautics and Space Administration  
 Cleveland, Ohio, October 11, 1985

## Appendix A

### List of Code Identifiers

Engineering symbol	Fortran symbol code	Comment
$A_{cx}$	ACX	composite axial stiffness; generated in subroutine GPCFD2
$A_{cx}^R$	RAC	reduced axial stiffness; computed in subroutine GPCFD2
BIDE	Boolean	true if interply effects are included; input
$C_{cx}$	CPC	composite coupling stiffness; generated in subroutine GPCFD2
$C_{e1}$	RESF	string with force variables in BLOCK DATA
$C_{e2}$	DISP	string with displacement variables in BLOCK DATA
COMSAT	Boolean	true if COMSA is executed; input
CSANB	Boolean	true if membrane and bending symmetry exists; input
$D_{cx}$	FTC	composite flexural rigidities; generated in subroutine GPCFD2
$D_{cx}^R$	RDC	reduced bending rigidities; computed in subroutine GPCFD2
$D_f$	DIAF	filament equivalent diameter; input
$D_v$	DISV, DISVI	displacement vectors; DISVI is either read in main program, or is generated in subroutine COMSA
$E_f, E_{cf}$	ECF	filament elastic constants; input
$E_{f11, f11, m11}$	EF11, EL11, EM11	filament, ply, and matrix normal moduli; filament and matrix moduli input
$G_{f12, f12, m11}$	EF12, EL12, EM12	filament, ply, and matrix shear moduli; filament and matrix shear moduli input
$E_p, E_{cp}$	ECL	ply elastic constants; generated in subroutine INHYD
$E_m, E_{cm}$	ECM	matrix elastic constants; generated in subroutine INHYD
$H_j$	PL(9,1)	interply distortion energy coefficient; generated in main program
$H_{kc}$	CHK	array of constituent heat conductivities; input
$h_c$	HHC	composite heat capacity stored in PC(18) and PC(54)
$i, j$	I, J	index; generally ply or interply
$K_{cxx, cyy, cxy}$	HK11, 22, 33	composite two-dimensional heat conductivities in PC(51) to PC(53)
$K_{c11, c22, c33}$	HK11, 22, 33	composite three-dimensional heat conductivities along the material axes in, PC(15) to PC(17)
$K_{f, v}$	KF, V	apparent fiber and void volume ratios; input
$K_{f11, f11, m11}$	CHK	see $H_{kc}$
$K_{1xx, 1yy, 1xy}$	XX1, XK2, XK3	stress concentration factors generated in STRCNF
$k_{f, m}$	KFB, MB	actual fiber and matrix volume ratios
$k_{f, v, t}$	KFL, VL	ply apparent fiber and void volume ratios
$L_{sc}$	LSC	array of limiting conditions; input
$M_{cx}$	MBS	applied moment; input
$M_{cT, x}$	MSDT	thermal moments; generated in GPCFD2
$M_{cM, x}$	MSDH	hygral moments; generated in GPCFD2
$m$	M	load condition index
$N_{cx}$	NBS	applied membrane loads; input
$N_{cM, x}$	NSDH	hygral force; generated in GPCFD2
$N_{cT, x}$	NSDT	thermal force; generated in GPCFD2
$N_f$	NFPE	number of filaments per end; input
$N_t$	NL	number of plies; input

$N_L$	NLC	number of load conditions; input
$N_{ms}$	NMS	number of material systems; input
$N_{pc}$	NPC	string PROPC length; input
$N_{pl}$	NPL	string PROP length; input
NONUDF	Boolean	T (true) if Poisson's ratio difference chart is to be suppressed
$P_c$	PC	composite properties array; generated in GACD3 and GPCFD2
$P_{cp}$	PROPC	string PROPC; composite property identifiers in GDCFD2
$P_l$	PL	ply property array; portions generated in all parts of the program
$P_p$	PROP	string PROP; ply properties identifiers in main program
$Q_{f,i,p,r,s}$	QF,I,P,R,S	indices to print out string PROP
$R$	R	transformation matrix; GACD3, GPCFD2, and COMSA
RINDV	Boolean	T (true) if displacements are read in; input
$S_{n17,etc.}$	PL(51) to PL(59,1)	ply limit stresses; generated in GLLSC
$t_l$	TL	ply thickness; input
$w_{cb}$	$w_{xx}$	composite local curvatures relative to the structural axes
$\alpha_c$	CTE	composite coefficient of thermal expansion; three-dimensional in PC(12) to PC(14), two-dimensional in PC(48) to PC(50)
$\alpha_{f,l,m}$	VAF,AL,AM	filament, ply, and matrix thermal coefficients of expansion; input
$\beta_{e,m}$	VCF	correlation factors for ply thermoelastic properties and strain magnification factors; set to unity in COMSA
$\beta_h$	BTA	correlation factors for ply heat conductivity; set to unity in COMSA
$\beta_s$	BET	correlation factors for ply strength; set to unity in COMSA
$\delta_l$	PL(8,1)	interply layer thickness; generated in INHYD
$\epsilon_{cr}$	UX	reference plane membrane strain; solved in terms of $N_{cr}$ or input
$\epsilon_l$	EPS, PL(74) to PL(66,1)	ply strains; generated in COMSA
$\theta_{cs}$	THCS	angle between composite material and structural axes; input
$\theta_g, \theta_L$	THLC	angle between ply material and composite axes; input
$\nu_{12}, n_2, m_{12}$	NUF12, L12, M12	filament, ply, and matrix Poisson's ratios; input
$\pi$	PIE	constant; input
$\rho_{f,m,l}$	RHOF, M, L	filament and matrix weight density; input and generated in FIBMT, HTM, and COMPP
$\sigma_f, \sigma_m$	SF, SM	microstresses in fibers and matrices generated in MCRSTR
$l_i$	XPL, XSL, YPL, YSL	boundary zone decay length; generated in the main program and paired to EDGSTR
$\sigma_l$	SIGL, PL(67) to PL(69,1)	ply stress; generated in COMSA

### Item 3

#### INPUT DATA ECHO

FOUR PLY SYMMETRIC LAMINATE. ICAN SAMPLE INPUT DATA.

```

STDATA      4      1      2
T
F
F
F
T
PLY      1      1      70.00      70.0
PLY      2      2      70.00      70.0
PLY      3      2      70.00      70.0
PLY      4      1      70.00      70.0
MATCHDAS--IMLS      .55      .02      AS--IMLS      0.0
MATCHDASOLAMPMS      .55      .01      AS--IMMS      0.4
PLOAD 1000.      0.0      0.0      0.0
PLOAD 0.0      0.0      0.0
PLOAD 0.0      0.0
COMSAT      CSAMB      SIDE      RINDB      NONUDF
T          F          F          F          T
          0.0      0.0      0.0      0.0
          90.0      0.0
          90.0      0.0
          0.0      0.0
          .97      .03
          .57      .01
          NX,NY,NXY,THCS
          NX,NY,MXY
          DMX/0X,DY/0Y,PRS3
  
```

### Item 4

#### SUMMARY OF INPUT DATA

FOUR PLY SYMMETRIC LAMINATE. ICAN SAMPLE INPUT DATA.

```

- - - CASE CONTROL DECK - - -
NUMBER OF LAYERS      NL = 4
NUMBER OF LOADING CONDITIONS      NLC = 1
NUMBER OF MATERIAL SYSTEMS      NMS = 2
  
```

```

COMSAT      CSAMB      SIDE      RINDB      NONUDF
T          F          F          F          T
  
```

```

- - - LAMINATE CONFIGURATION - - -
-----
PLY      NO      MID      DELTAT      DELTAM      THETA      T-MESS
-----
PLY      1      1      0.000      0.0X      0.0      0.010
PLY      2      2      0.000      0.0X      90.0      0.005
PLY      3      2      0.000      0.0X      90.0      0.005
PLY      4      1      0.000      0.0X      0.0      0.010
-----
  
```

```

- - - COMPOSITE MATERIAL SYSTEMS - - -
-----
MATCHD      MID      PRIMARY      VFP      VVP      SECONDARY      VSC      VFS      VVS
MA 70      1      AS--IMLS      0.55      0.02      AS--IMLS      0.00      0.97      0.03
MA 90      2      SOLAMPMS      0.55      0.01      AS--IMMS      0.40      0.57      0.01
-----
  
```

```

- - - LOADING CONDITIONS - - -
PRESCRIBED LOADS FOR THE LOAD CONDITION 1
INPLANE LOADS      NX = 1000.0000 LB/IN
                   NY = 0.0000 LB/IN
                   NXY = 0.0000 LB/IN
BENDING LOADS      MX = 0.0000 LB.IN/IN
                   MY = 0.0000 LB.IN/IN
                   MXY = 0.0000 LB.IN/IN
TRANSVERSE LOADS  DMX/0X = 0.0000 LB/IN
                   DMY/0Y = 0.0000 LB/IN
TRANSVERSE PRESSURE      PU = 0.0000 LB/SQ. IN.
TRANSVERSE PRESSURE      PL = 0.0000 LB/SQ. IN.
  
```

## Appendix B

### Sample Input/Output

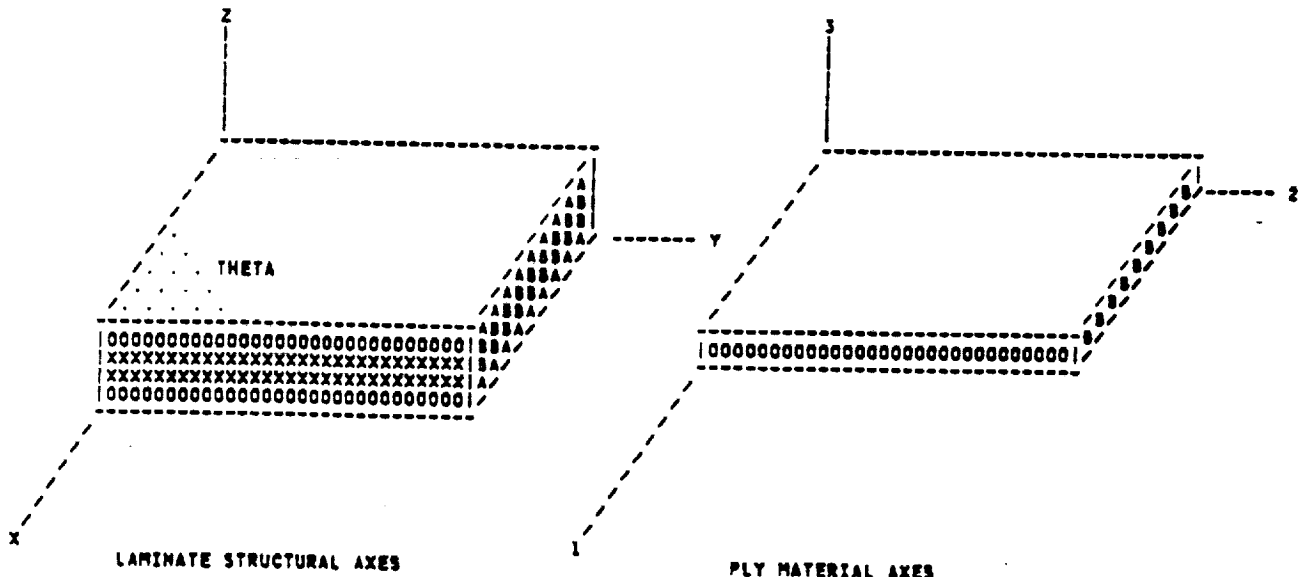
**Item 1**

I C A N

[illegible]

**Item 2**

## I C A N: COORDINATE SYSTEMS





# Item 5(a)

--> CONSTITUENT PROPERTIES: ECHO FROM DATA BANK. <--

PRIMARY FIBER PROPERTIES: AS-- FIBER			
1	ELASTIC MODULI	ETP1	0.3100E 08
2		ETP2	0.2000E 07
3	SHEAR MODULI	GFP12	0.2000E 07
4		GFP23	0.1000E 07
5	POISSON'S RATIO	NUFP12	0.2000E 00
6		NUFP23	0.2500E 00
7	THERM. EXP. COEF.	CTEFP1	-0.5500E-04
8		CTEFP2	0.5600E-03
9	DENSITY	RHOF1	0.6300E-01
10	NO. OF FIBERS/END	NFP	0.1000E 03
11	FIBER DIAMETER	DIFP	0.3000E-03
12	HEAT CAPACITY	CFPC	0.1700E 00
13	HEAT CONDUCTIVITY	KFP1	0.5000E 03
14		KFP2	0.5000E 02
15		KFP3	0.5000E 02
16	STRENGTHS	SFPT	0.4000E 06
17		SFPC	0.4000E 06

PRIMARY MATRIX PROPERTIES: IMLS MATRIX. DRY RT. PROPERTIES.

1	ELASTIC MODULUS	EMP	0.9000E 06
2	SHEAR MODULUS	GNP	0.1773E 06
3	POISSON'S RATIO	MUTP	0.4100E 00
4	THERM. EXP. COEF.	CTEMP	0.5700E-04
5	DENSITY	RHOMP	0.4400E-01
6	HEAT CAPACITY	CMPC	0.2500E 00
7	HEAT CONDUCTIVITY	KMP	0.1250E 01
8	STRENGTHS	SMPT	0.7000E 04
9		SMPC	0.2100E 03
10		SMPS	0.7000E 04
11	MOISTURE COEF	BTAMP	0.4000E-02
12	DIFFUSIVITY	DIFMP	0.2000E-03

# Item 5(b)

PRIMARY COMPOSITE PROPERTIES: 55/ 43 AS--/IMLS

BASED ON MICROMECHANICS OF ENTRAPLY HYBRID COMPOSITES: ELASTIC AND THERMAL PROPERTIES.

FIBER VOLUME RATIO - 0.550      MATRIX VOLUME RATIO - 0.430      VOID VOLUME RATIO - 0.020  
VOID CONDUCTIVITY - 0.22499990E 08

1	ELASTIC MODULI	EPC1	0.1726E 08
2		EPC2	0.1127E 07
3		EPC3	0.1127E 07
4	SHEAR MODULI	GPC12	0.5470E 06
5		GPC23	0.3238E 06
6		GPC13	0.5470E 06
7	POISSON'S RATIO	NUPC12	0.2945E 00
8		NUPC23	0.4021E 00
9		NUPC13	0.2945E 00
10	THERM. EXP. COEF.	CTEPC1	0.1418E-04
11		CTEPC2	0.2464E-04
12		CTEPC3	0.2464E-04
13	DENSITY	RHOPC	0.5443E-01
14	HEAT CAPACITY	CPC	0.1991E 00
15	HEAT CONDUCTIVITY	KPC1	0.3195E 03
16		KPC2	0.3702E 01
17		KPC3	0.3702E 01
18	STRENGTHS	SPC1T	0.2228E 06
19		SFC1C	0.8744E 05
20		SPC2T	0.5006E 04
21		SPC2C	0.1502E 05
22		SPC12	0.5126E 04
23	MOIST. DIFFUSIVITY	DPC1	0.8600E-04
24		DPC2	0.5168E-04
25		DPC3	0.5168E-04
26	MOIST. EXP. COEF.	BTAPC1	0.4981E-04
27		BTAPC2	0.1452E-02
28		BTAPC3	0.1452E-02
29	FLEXURAL MODULI	EPC1F	0.1726E 08
30		EPC2F	0.1127E 07
31	STRENGTHS	SFC23	0.3983E 04
32		SPC1F	0.1572E 06
33		SPC2F	0.9387E 04
34		SPC5B	0.7489E 04
35	PLY THICKNESS	TPC	0.5000E-02
36	INTERPLY THICKNESS	PLPC	0.5850E-04
37	INTERFIBER SPACING	PLPCS	0.5850E-04

# Item 5(c)

--> CONSTITUENT PROPERTIES: ECHO FROM DATA BANK. <--

PRIMARY FIBER PROPERTIES: SOLA FIBER

1	ELASTIC MODULI	EPF1	0.1240E 08
2		EPF2	0.1240E 08
3	SHEAR MODULI	GPF12	0.5170E 07
4		GPF23	0.5170E 07
5	POISSON'S RATIO	NUFP12	0.2800E 00
6		NUFP23	0.2800E 00
7	THERM. EXP. COEF.	CTEFP1	0.2800E-05
8		CTEFP2	0.2800E-05
9	DENSITY	RHOPF	0.9000E-01
10	NO. OF FIBERS/END	NFP	0.2040E 03
11	FIBER DIAMETER	DIFP	0.3600E-03
12	HEAT CAPACITY	CFPC	0.1700E 00
13	HEAT CONDUCTIVITY	KFP1	0.7500E 01
14		KFP2	0.7500E 01
15		KFP3	0.7500E 01
16	STRENGTHS	SFP1	0.5600E 04
17		SFPC	0.3000E 06

PRIMARY MATRIX PROPERTIES: MMMS MATRIX. DRY RT. PROPERTIES.

1	ELASTIC MODULUS	EMP	0.7500E 06
2	SHEAR MODULUS	GMP	0.2778E 06
3	POISSON'S RATIO	NUMP	0.3500E 00
4	THERM. EXP. COEF.	CTEMP	0.4000E-04
5	DENSITY	RHOMP	0.4500E-01
6	HEAT CAPACITY	CMPC	0.2500E 00
7	HEAT CONDUCTIVITY	KMP	0.1250E 01
8	STRENGTHS	SMPT	0.2000E 05
9		SMPC	0.5000E 05
10		SMPS	0.1500E 05
11	MOISTURE COEF	BTAMP	0.4000E-02
12	DIFFUSIVITY	DIFMP	0.2000E-03

# Item 5(d)

PRIMARY COMPOSITE PROPERTIES: 55/ 44 SOLA/MMMS

BASED ON MICROMECHANICS OF INTRAPLY HYBRID COMPOSITES: ELASTIC AND THERMAL PROPERTIES.

FIBER VOLUME RATIO - 0.550      MATRIX VOLUME RATIO - 0.440      VOID VOLUME RATIO - 0.0  
VOID CONDUCTIVITY - 0.22499990E 00

1	ELASTIC MODULI	EPC1	0.7150E 07
2		EPC2	0.2473E 07
3		EPC3	0.2473E 07
4	SHEAR MODULI	GPC12	0.9314E 06
5		GPC23	0.9379E 06
6		GPC13	0.9314E 06
7	POISSON'S RATIO	NUPC12	0.2675E 00
8		NUPC23	0.3778E 00
9		NUPC13	0.2675E 00
10	THERM. EXP. COEF.	CTEPC1	0.4400E-05
11		CTEPC2	0.1500E-04
12		CTEPC3	0.1500E-04
13	DENSITY	RHOPC	0.6930E-01
14	HEAT CAPACITY	CPC	0.1929E 00
15	HEAT CONDUCTIVITY	KPC1	0.4675E 01
16		KPC2	0.2750E 01
17		KPC3	0.2750E 01
18	STRENGTHS	SPC1T	0.2876E 06
19		SPC1C	0.1730E 06
20		SPC2T	0.1256E 05
21		SPC2C	0.3140E 05
22		SPC12	0.1047E 05
23	MOIST. DIFFUSIVITY	DPC1	0.8800E-04
24		DPC2	0.5168E-04
25		DPC3	0.5168E-04
26	MOIST. EXP. COEF.	BTAPC1	0.1846E-03
27		BTAPC2	0.1379E-02
28		BTAPC3	0.1379E-02
29	FLEXURAL MODULI	EPC1F	0.7150E 07
30		EPC2F	0.2473E 07
31	STRENGTHS	SPC23	0.6510E 04
32		SPC1F	0.2359E 06
33		SPC2F	0.2243E 05
34		SPC3F	0.1579E 05
35	PLY THICKNESS	TPC	0.5000E-02
36	INTERPLY THICKNESS	PLPC	0.7020E-04
37	INTERFIBER SPACING	PLPCS	0.7020E-04

# Item 5(e)

--> CONSTITUENT PROPERTIES: ECHO FROM DATA BANK. <--

SECONDARY FIBER PROPERTIES: AS-- FIBER

1	ELASTIC MODULI	EFS1	0.3100E 09
2		EFS2	0.2000E 07
3	SHEAR MODULI	GFS12	0.2000E 07
4		GFS23	0.1000E 07
5	POISSON'S RATIO	MUFS12	0.2000E 00
6		MUFS23	0.2500E 00
7	THERM. EXP. COEF.	CTEFS1	-0.5500E-06
8		CTEFS2	0.5600E-05
9	DENSITY	RHOFS	0.6300E-01
10	NO. OF FIBERS/END	NFS	0.1000E 09
11	FIBER DIAMETER	DIFS	0.3000E-03
12	HEAT CAPACITY	CFSC	0.1700E 00
13	HEAT CONDUCTIVITY	KFS1	0.5000E 03
14		KFS2	0.5000E 02
15		KFS3	0.5000E 02
16	STRENGTHS	STST	0.4000E 06
17		STSC	0.4000E 06

SECONDARY MATRIX PROPERTIES: IMHS MATRIX. DRY RT. PROPERTIES.

1	ELASTIC MODULUS	EMS	0.5000E 06
2	SHEAR MODULUS	GHS	0.1052E 06
3	POISSON'S RATIO	MUMS	0.3500E 00
4	THERM. EXP. COEF.	CTEMS	0.3600E-04
5	DENSITY	RHOMS	0.4400E-01
6	HEAT CAPACITY	CMSC	0.2500E 00
7	HEAT CONDUCTIVITY	KMS	0.1250E 01
8	STRENGTHS	SHST	0.1500E 05
9		SHSC	0.3500E 05
10		SHSS	0.1500E 05
11	MOISTURE COEF	BTAMS	0.4000E-02
12	DIFFUSIVITY	DIFMS	0.2000E-03

# Item 5(f)

SECONDARY COMPOSITE PROPERTIES: 57/ 42 AS--/IMHS

BASED ON MICROMECHANICS OF INTERPLY HYBRID COMPOSITES: ELASTIC AND THERMAL PROPERTIES.

FIBER VOLUME RATIO - 0.570      MATRIX VOLUME RATIO - 0.420      VOID VOLUME RATIO - 0.010  
VOID CONDUCTIVITY - 0.22499990E 00

1	ELASTIC MODULI	ESC1	0.1788E 08
2		ESC2	0.1153E 07
3		ESC3	0.1153E 07
4	SHEAR MODULI	GSC12	0.5000E 06
5		GSC23	0.3450E 06
6		GSC13	0.5000E 06
7	POISSON'S RATIO	MUSC12	0.2645E 00
8		MUSC23	0.4294E 00
9		MUSC13	0.2645E 00
10	THERM. EXP. COEF.	CTESC1	-0.1280E-06
11		CTESC2	0.1605E-04
12		CTESC3	0.1605E-04
13	DENSITY	RHOSC	0.5439E-01
14	HEAT CAPACITY	CSC	0.1972E 00
15	HEAT CONDUCTIVITY	KSC1	0.3311E 03
16		KSC2	0.3910E 01
17		KSC3	0.3910E 01
18	STRENGTHS	SSC1T	0.2307E 06
19		SSC1C	0.1560E 06
20		SSC2T	0.1026E 05
21		SSC2C	0.2394E 05
22		SSC12	0.9369E 04
23	MOIST. DIFFUSIVITY	DSC1	0.8400E-04
24		DSC2	0.4900E-04
25		DSC3	0.4910E-04
26	MOIST. EXP. COEF.	BTASC1	0.4650E-04
27		BTASC2	0.1319E-02
28		BTASC3	0.1319E-02
29	FLEXURAL MODULI	ESC1F	0.1788E 08
30		ESC2F	0.1153E 07
31	STRENGTHS	SSC23	0.7424E 04
32		SSC1F	0.2334E 06
33		SSC2F	0.1794E 05
34		SSC3F	0.1405E 05
35	PLY THICKNESS	TSC	0.5000E-02
36	INTERPLY THICKNESS	PLSC	0.5215E-04
37	INTERFIBER SPACING	PLSCS	0.5215E-04

Item 5(g)

HYBRID COMPOSITE PROPERTIES; 60/40 SGLA/HMHS/AS--/IMHS  
 BASED ON MICROMECHANICS OF INTRAPLY HYBRID COMPOSITES: ELASTIC AND THERMAL PROPERTIES

PRIMARY COMPOSITE VOLUME RATIO - 0.600

SECONDARY COMPOSITE VOLUME RATIO - 0.40

1	ELASTIC MODULI	EHC1	0.1144E 08
2		EHC2	0.1696E 07
3		EHC3	0.1945E 07
4	SHEAR MODULI	GHC12	0.7551E 06
5		GHC23	0.4561E 06
6		GHC13	0.7941E 06
7	POISSON'S RATIO	NUHC12	0.2663E 00
8		NUHC23	0.3985E 00
9		NUHC13	0.2689E 00
10	THERM. EXP. COEF.	CTEHC1	0.1603E-05
11		CTEHC2	0.1601E-04
12		CTEHC3	0.1634E-04
13	DENSITY	RHOHC	0.6334E-01
14	HEAT CAPACITY	CHC	0.1943E 00
15	HEAT CONDUCTIVITY	KHC1	0.1352E 03
16		KHC2	0.2305E 01
17		KHC3	0.2305E 01
18	STRENGTHS	SHC1T	0.2168E 06
19		SHC1C	0.1665E 06
20		SHC2T	0.9915E 04
21		SHC2C	0.2314E 05
22		SHC12	0.1195E 05
23	MOIST. DIFFUSIVITY	DHC1	0.8736E-04
24		DHC2	0.5117E-04
25		DPC3	0.5117E-04
26	MOIST. EXP. COEF.	BTAHC1	0.9858E-04
27		BTAHC2	0.8565E-03
28		BTAHC3	0.1455E-02
29	FLEXURAL MODULI	EHC1F	0.1144E 08
30		EHC2F	0.1696E 07
31	STRENGTHS	SHC23	0.1019E 05
32		SHC1F	0.2355E 06
33		SHC2F	0.1735E 05
34		SHCSB	0.1793E 05
35	PLY THICKNESS	THC	0.5000E-02
36	INTERPLY THICKNESS	PLHC	0.5215E-04
37	INTERFIBER SPACING	PLHCS	0.5215E-04
38	FIBER VOL. RATIO	VFH	0.5580E 00
39	MOISTURE CONTENT	M	0.0000
40	MATRIX VOL. RATIO	VMH	0.4320E 00

# Item 6

## 3-D COMPOSITE STRAIN STRESS TEMPERATURE MOISTURE RELATIONS - STRUCTURAL AXES

	-1-	-2-	-3-	-4-	-5-	-6-	-7-	-8-
1	0.6976E-07	-0.9952E-08	-0.2727E-07	0.0000	0.0000	0.3255E-13	0.1102E-03	0.1009E-03
2	-0.5952E-08	0.1962E-06	-0.8445E-07	0.0000	0.0000	-0.1444E-11	0.3009E-05	0.1370E-03
3	-0.2727E-07	-0.8403E-07	0.3614E-06	0.0000	0.0000	0.4682E-12	0.2039E-04	0.1859E-02
4	0.0000	0.0000	0.0000	0.2139E-03	0.5229E-12	0.0000	0.0000	0.0000
5	0.0000	0.0000	0.0000	0.5229E-12	0.1935E-03	0.0000	0.0000	0.0000
6	0.3255E-13	-0.1444E-11	0.4682E-12	0.0000	0.0000	0.1622E-05	-0.4330E-10	-0.2425E-05

## 3-D COMPOSITE STRESS STRAIN RELATIONS - STRUCTURAL AXES

	-1-	-2-	-3-	-4-	-5-	-6-
1	0.1473E 08	0.8093E 06	0.8377E 06	0.0000	0.0000	0.8960E-01
2	0.8093E 06	0.5499E 07	0.8703E 06	0.0000	0.0000	0.4587E 01
3	0.8377E 06	0.8703E 06	0.1953E 07	0.0000	0.0000	-0.3403E-01
4	0.0000	0.0000	0.0000	0.4476E 06	-0.1263E 00	0.0000
5	0.0000	0.0000	0.0000	-0.1263E 00	0.5167E 06	0.0000
6	0.8960E-01	0.4587E 01	-0.3403E-01	0.0000	0.0000	0.6164E 06

MAT9 CARD FOR MSC/NASTRAN SOLID ELEMENTS

511.012.013.014.015.016.022.023.024.025.026.033.034.035.036.044.045.046.053.054.044  
0.14731064E 08 0.80927925E 06 0.83770038E 06 0.89597344E-01 0.00000000 0.00000000 0.34987690E 07 0.87032406E 06  
0.45863379E 01 0.00000000 0.00000000 0.19531170E 07 -0.34027349E-01 0.00000000 0.00000000 0.61636813E 06  
0.00000000 0.00000000 0.46757319E 06 -0.12633294E 00 0.51670636E 06

# Item 7

## COMPOSITE PROPERTIES

=====

COMPOSITE PROPERTIES - VALID ONLY FOR CONSTANT TEMPERATURE THROUGH THICKNESS

LINE 1 TO 31 3-D COMPOSITE PROPERTIES ABOUT MATERIAL AXES

LINE 33 TO 62 2-D COMPOSITE PROPERTIES ABOUT STRUCTURAL AXES

=====

1	RHGC	0.9740E-01	32	SZDEC	0.0000
2	TC	0.3000E-01	33	CC11	0.1213E 08
3	CC11	0.1473E 08	34	CC12	0.3746E 06
4	CC12	0.8093E 06	35	CC13	0.1067E 00
5	CC13	0.8377E 06	36	CC22	0.4410E 07
6	CC22	0.5499E 07	37	CC23	0.4058E 01
7	CC23	0.8703E 06	38	CC33	0.6164E 06
8	CC33	0.1953E 07	39	EC11	0.1212E 08
9	CC44	0.4676E 06	40	EC22	0.4598E 07
10	CC55	0.5167E 06	41	EC12	0.6164E 06
11	CC66	0.6164E 06	42	NUC12	0.6126E-01
12	CYE11	0.1102E-05	43	NUC21	0.3084E-01
13	CYE22	0.5803E-05	44	CSM13	-0.3613E-06
14	CYE33	0.2839E-06	45	CSM31	-0.1839E-07
15	NK11	0.2138E 03	46	CSM23	0.4874E-05
16	NK22	0.4755E 02	47	CSM32	0.8811E-06
17	NK33	0.3080E 01	48	CYE11	0.1102E-05
18	HMC	0.1975E 00	49	CYE22	0.5803E-05
19	EC11	0.1433E 08	50	CYE12	-0.4534E-10
20	EC22	0.5898E 07	51	NK11	0.2138E 03
21	EC33	0.1781E 07	52	NK22	0.4755E 02
22	EC23	0.4676E 06	53	NK12	-0.5783E-04
23	EC31	0.5167E 06	54	HMC	0.1975E 00
24	EC12	0.6164E 06	55	DPC11	0.1575E-02
25	NUC12	0.8531E-01	56	DPC22	0.1576E-02
26	NUC21	0.3034E-01	57	DPC33	0.2000E 05
27	NUC13	0.3900E 00	58	DPC12	0.0000
28	NUC31	0.4836E-01	59	BTAC11	0.1009E-03
29	NUC23	0.4326E 00	60	BTAC22	0.1370E-03
30	NUC32	0.1511E 00	61	BTAC33	0.1859E-02
31	ZCCC	0.1500E-01	62	BTAC12	-0.2425E-05

# Item 8

FORCES	FORCE DISPLACEMENT RELATIONS							DISPL	T-FORCES	N-
UX	0.3644E 06	0.1124E 05	0.3201E-02	0.0000	0.0000	-0.7276E-11	0.0000	UX	0.0000	
UY	0.1124E 05	0.1383E 06	0.1217E 00	0.0000	-0.1373E-03	-0.2619E-09	0.0000	UY	0.0000	
UXY	0.3201E-02	0.1217E 00	0.1849E 05	-0.7276E-11	-0.2619E-09	0.0000	0.0000	UXUY	0.0000	
MX	0.0000	0.0000	-0.7276E-11	0.3776E 02	0.7610E 00	0.2667E-07	0.0000	MX	0.0000	
MY	0.0000	-0.1373E-03	-0.2619E-09	0.7610E 00	0.3419E 01	0.1014E-05	0.0000	MY	0.0000	
MAX	-0.7276E-11	-0.2619E-09	0.0000	0.2667E-07	0.1014E-05	0.1248E 01	0.0000	MAX	0.0000	

# Item 9

## REDUCED STIFFNESS MATRIX

0.3644E 06 0.1124E 05 0.3200E-02  
0.1124E 05 0.1383E 06 0.1216E 00  
0.3200E-02 0.1216E 00 0.1849E 05

## REDUCED BENDING RESIDUES

0.3776E 02 0.7610E 00 0.2667E-07  
0.7610E 00 0.3419E 01 0.1013E-05  
0.2667E-07 0.1013E-05 0.1248E 01

# Item 10

## SOME USEFUL DATA FOR P.E. ANALYSIS

COMPOSITE THICKNESS FOR P.E. ANALYSIS = 0.3000E-01

PROPERTIES FOR P.E. ANALYSIS E11,E12,E13,E22,E23,E33 PROPERTIES SCALED BY 1000-6  
0.32533E-01 -0.67069E-02 -0.29839E-07 0.21747E 00 0.14294E-05 0.16224E 01

BENDING EQUIVALENT PROPERTIES MUXY, MCYX, EXX, EGY, GCYX  
0.22261E 00 0.28153E-01 0.16708E 00 0.15126E 07 0.35473E 06

MASTRAN MEMBRANE EQUIVALENT ELASTIC COEFFICIENTS G11,G12,G13,G22,G23,G33  
0.12147E 00 0.37445E 06 0.18669E 00 0.44899E 07 0.44351E 01 0.61637E 06

MASTRAN BENDING EQUIVALENT ELASTIC COEFFICIENTS G11,G12,G13,G22,G23,G33  
0.16734E 00 0.33824E 06 0.11854E-01 0.15194E 07 0.45054E 00 0.35473E 06

# Item 11

DISP.	DISPLACEMENT FORCE RELATIONS							COMBINED FORCES
1	0.2791E-02	0.2791E-09	-0.2234E-06	0.9946E-12	0.1818E-12	-0.9821E-11	-0.2356E-16	0.1000E 04
2	-0.2234E-03	-0.2234E-06	0.7249E-05	-0.4763E-10	-0.3695E-11	0.2923E-09	0.1283E-14	0.0000
3	0.9946E-09	0.9946E-12	-0.4763E-10	0.3468E-04	-0.3466E-16	0.2237E-14	-0.1181E-19	0.0000
4	0.1818E-09	0.1818E-12	-0.3695E-11	-0.3466E-16	0.2660E-01	-0.5922E-02	0.4241E-08	0.0000
5	-0.9821E-08	-0.9821E-11	0.2923E-09	0.2237E-14	-0.5922E-02	0.2936E 00	-0.2385E-06	0.0000
6	-0.2356E-13	-0.2356E-16	0.1283E-14	-0.1181E-19	0.4241E-08	-0.2385E-06	0.8012E 00	0.0000

NOTE: THE DISPLACEMENTS ARE REFERENCE PLANE MEMBRANE STRAINS (UX, UY, UXUY) AND CURVATURES (MX, MY, MXY)

## Item 12

## PLY HYGROTHERMOMECHANICAL PROPERTIES / RESPONSE

FOR LOAD CONDITIONS  
 MEMBRANE LOADS MES(X,Y,XY-M) ARE 1000. 0. 0.  
 BENDING LOADS MES(X,Y,XY-M) ARE 0. 0. 0.  
 QXZ,QYZ AND APPLIED PRESSURES ARE 0. 0. 0.  
 NOTE : NO MOISTURE OR TEMPERATURE

LAYER PROPERTIES, ROWS-PROPERTY, COLUMNS-LAYER

PLY NUMBER	1	2	3	4
MATERIAL SYSTEM	AS--/TMS	SGLA/TMS	SGLA/TMS	AS--/TMS
ORIENTATION	AS--/TMS	AS--/TMS	AS--/TMS	AS--/TMS
	0.0	90.0	90.0	0.0
1 KV	0.2800E-01	0.1000E-01	0.1000E-01	0.2800E-01
2 KP	0.5500E 00	0.5500E 00	0.5500E 00	0.5500E 00
3 KPB	0.5390E 00	0.5524E 00	0.5524E 00	0.5390E 00
4 KM	0.4500E 00	0.4420E 00	0.4420E 00	0.4500E 00
5 KMB	0.4410E 00	0.4376E 00	0.4376E 00	0.4410E 00
6 PHOL	0.9443E-01	0.6334E-01	0.6334E-01	0.9443E-01
7 TL	0.1000E-01	0.5000E-02	0.5000E-02	0.1000E-01
8 DELTA	0.5050E-04	0.5215E-04	0.5215E-04	0.5050E-04
9 ILDC	0.0000	0.0000	0.0000	0.0000
10 ZB	0.5000E-02	0.1250E-01	0.1750E-01	0.2500E-01
11 ZGC	-0.1000E-01	-0.2500E-02	0.2500E-02	0.1000E-01
12 THCS	0.0000	0.0000	0.0000	0.0000
13 THLC	0.0000	0.1571E 01	0.1571E 01	0.0000
14 THLS	0.0000	0.1571E 01	0.1571E 01	0.0000
15 SC11	0.2101E 00	0.1323E 00	0.1323E 00	0.2101E 00
16 SC12	0.7797E 06	0.8684E 06	0.8684E 06	0.7797E 06
17 SC13	0.7797E 06	0.8684E 06	0.8684E 06	0.7797E 06
18 SC22	0.1631E 07	0.2166E 07	0.2166E 07	0.1631E 07
19 SC23	0.8713E 06	0.9537E 06	0.9537E 06	0.8713E 06
20 SC33	0.1774E 07	0.2307E 07	0.2307E 07	0.1774E 07
21 SC44	0.3238E 06	0.4561E 06	0.4561E 06	0.3238E 06
22 SC39	0.5470E 06	0.7551E 06	0.7551E 06	0.5470E 06
23 SC66	0.5470E 06	0.7551E 06	0.7551E 06	0.5470E 06
24 CTE11	0.1418E-06	0.1603E-05	0.1603E-05	0.1418E-06
25 CTE22	0.2464E-04	0.1601E-04	0.1601E-04	0.2464E-04
26 CTE33	0.2464E-04	0.1601E-04	0.1601E-04	0.2464E-04
27 HK11	0.3195E 03	0.1352E 03	0.1352E 03	0.3195E 03
28 HK22	0.3702E 01	0.2305E 01	0.2305E 01	0.3702E 01
29 HK33	0.3702E 01	0.2305E 01	0.2305E 01	0.3702E 01
30 HCL	0.1991E 00	0.1943E 00	0.1943E 00	0.1991E 00
31 EL11	0.1726E 08	0.1144E 08	0.1144E 08	0.1726E 08
32 EL22	0.1127E 07	0.1696E 07	0.1696E 07	0.1127E 07
33 EL33	0.1127E 07	0.1696E 07	0.1696E 07	0.1127E 07
34 GL23	0.3238E 06	0.4561E 06	0.4561E 06	0.3238E 06
35 GL13	0.5470E 06	0.7551E 06	0.7551E 06	0.5470E 06
36 GL12	0.5470E 06	0.7551E 06	0.7551E 06	0.5470E 06
37 MUL12	0.2945E 00	0.2663E 00	0.2663E 00	0.2945E 00
38 MUL23	0.1922E-01	0.3947E-01	0.3947E-01	0.1922E-01
39 MUL13	0.2945E 00	0.2663E 00	0.2663E 00	0.2945E 00
40 MUL31	0.1922E-01	0.3947E-01	0.3947E-01	0.1922E-01
41 MUL23	0.4821E 00	0.3985E 00	0.3985E 00	0.4821E 00
42 MUL32	0.4821E 00	0.3985E 00	0.3985E 00	0.4821E 00
43 DPL1	0.8600E-04	0.8736E-04	0.8736E-04	0.8600E-04
44 DPL2	0.5168E-04	0.5117E-04	0.5117E-04	0.5168E-04
45 DPL3	0.5168E-04	0.5117E-04	0.5117E-04	0.5168E-04
46 BTAL1	0.4981E-04	0.9853E-04	0.9853E-04	0.4981E-04
47 BTAL2	0.1452E-02	0.8565E-03	0.8565E-03	0.1452E-02
48 BTAL3	0.1452E-02	0.1455E-02	0.1455E-02	0.1452E-02
49 ILHFC	0.0000	0.8405E 02	0.8405E 02	0.8405E 02
50 TEMPD	0.0000	0.0000	0.0000	0.0000
51 LSC11T	0.2228E 06	0.2168E 06	0.2168E 06	0.2228E 06
52 LSC11C	0.8764E 05	0.1665E 06	0.1665E 06	0.8764E 05
53 LSC11D	0.8764E 05	0.1665E 06	0.1665E 06	0.8764E 05
54 LSC22T	0.5006E 04	0.9915E 04	0.9915E 04	0.5006E 04
55 LSC22C	0.1502E 05	0.2314E 05	0.2314E 05	0.1502E 05
56 LSC12	0.5126E 04	0.1195E 05	0.1195E 05	0.5126E 04
57 LSC23	0.3983E 04	0.1019E 05	0.1019E 05	0.3983E 04
58 LSCC23	0.0000	0.6196E 05	0.1351E 06	0.4417E 05
59 LSCC13	0.0000	0.7303E 05	0.8147E 05	0.5230E 05
60 LSCDP	0.0000	0.4164E-03	0.3925E-03	0.4164E-03
61 KL12AB	0.9858E 00	0.9075E 00	0.9075E 00	0.9858E 00
62 MDEIE	0.9646E 00	0.7801E 00	0.7801E 00	0.9646E 00
63 RELROT	0.0000	0.1000E 01	0.1000E 01	0.1000E 01
64 EPS11	0.2751E-02	-0.2236E-03	-0.2236E-03	0.2751E-02
65 EPS22	-0.2236E-03	0.2751E-02	0.2751E-02	-0.2236E-03
66 EPS12	0.9946E-09	-0.8536E-08	-0.8536E-08	0.9946E-09
67 SIG11	0.4769E 05	-0.1329E 04	-0.1329E 04	0.4769E 05
68 SIG22	0.6647E 03	0.4614E 04	0.4614E 04	0.6647E 03
69 SIG12	0.5441E-03	-0.6445E-02	-0.6445E-02	0.5441E-03
70 DELFI	0.0000	-0.4765E-08	0.0000	0.4765E-08
71 HFC	0.1121E 01	0.6393E 00	0.6393E 00	0.1121E 01
72 MPCTGE	0.0000	0.0000	0.0000	0.0000
73 SIG13	0.0000	0.0000	0.0000	0.0000
74 SIG23	0.0000	0.0000	0.0000	0.0000
75 SIG33	0.0000	0.0000	0.0000	0.0000

# Item 13

## DETAILS OF POISSON RATIO MISMATCH

### POISSON'S RATIOS OF THE COMPOSITE

ANUCXY = 0.0013  
ANUCYX = 0.0308  
ANUCSX = -0.0000  
ANUCSY = 0.0000

NO.	THETA	ANULXY	ANULSX	ANULSY	POIDFM	POIDFS
1	0.0	0.2945	0.0000	0.0000	0.2132	-0.0000
2	90.0	0.0395	-0.0000	-0.0000	-0.0418	-0.0000
3	90.0	0.0395	-0.0000	-0.0000	-0.0418	-0.0000
4	0.0	0.2945	0.0000	0.0000	0.2132	-0.0000

# Item 14

## FREE EDGE STRESSES

PLY	THETA	SIGXX	SIGYY	SIGXY	YDCAT LENGTH	SIGZY	SIGZZ	SIGXZ
1	0.0	0.143E 01	0.199E-01	0.163E-07	0.119E 00	0.116E-01	0.337E-02	0.412E-00
1	0.0	0.143E 01	0.199E-01	0.163E-07	0.000	0.116E-01	0.337E-02	0.412E-00
2	90.0	0.138E 00	-0.399E-01	-0.326E-07	0.298E 00	-0.979E-00	0.801E-03	0.248E-00
2	90.0	0.138E 00	-0.399E-01	-0.326E-07	0.345E 00	-0.845E-00	0.290E-03	0.248E-00
3	90.0	0.138E 00	-0.399E-01	-0.326E-07	0.298E 00	-0.979E-00	0.801E-03	0.248E-00
3	90.0	0.138E 00	-0.399E-01	-0.326E-07	0.345E 00	-0.979E-00	0.290E-03	0.248E-00
4	0.0	0.143E 01	0.199E-01	0.163E-07	0.119E 00	0.116E-01	0.337E-02	0.412E-00
4	0.0	0.143E 01	0.199E-01	0.163E-07	0.000	0.116E-01	0.337E-02	0.412E-00

NOTE: THE INTERLAMINAR STRESSES ARE BETWEEN PLYS (I-1) AND (I).

NOTE: IF THE PLY NO IS REPEATED THEN THE SECOND ONE INDICATES STRESSES IN THE SECONDARY COMPOSITE.

NOTE: FOR ANGLE PLY LAMINATES SIGYY IS 0. CONSEQUENTLY SIGZY AND SIGZZ ARE COMPUTED AS ZERO.  
TO OBTAIN NONTRIVIAL SIGZY AND SIGZZ, ONE MUST SPECIFY A THIN INTERPLY LAYER.  
THE INTERPLY LAYER THICKNESS MAY BE OBTAINED FROM THE PLY PROPERTY TABLE.



## Item 15(n)

## MICROSTRESSES

FOR LOAD CONDITIONS  
 MEMBRANE LOADS  $MES(X,Y,XY-M)$  ARE 1000. 0. 0.  
 BENDING LOADS  $MBS(X,Y,XY-M)$  ARE 0. 0. 0.  
 $QXZ,QYZ$  AND APPLIED PRESSURES ARE 0. 0. 0.  
 NOTE: NO MOISTURE OR TEMPERATURE

(NOTE: ROWS-PROPERTY, COLUMNS-LAYER)

PLY NUMBER		1	2	3	4
MATERIAL SYSTEM		AS--/IMLS	SOLA/MPMS	SOLA/MPMS	AS--/IMLS
ORIENTATION		AS--/IMLS	AS--/IMMS	AS--/IMMS	AS--/IMLS
		0.0	90.0	90.0	0.0
1	SM1L	0.1381E 04	-0.8714E 02	-0.8714E 02	0.1381E 04
1	SM1L	0.0000	-0.5809E 02	-0.5809E 02	0.0000
2	SM1T	0.2669E 03	0.1534E 04	0.1534E 04	0.2669E 03
2	SM1T	0.0000	0.1561E 04	0.1561E 04	0.0000
3	SP1L	0.8564E 05	-0.1441E 04	-0.1441E 04	0.8564E 05
3	SP1L	0.0000	-0.3602E 04	-0.3602E 04	0.0000
4	SP1T	-0.2183E 03	-0.4087E 03	-0.4087E 03	-0.2183E 03
4	SP1T	0.0000	-0.2406E 04	-0.2406E 04	0.0000
5	SM2AL	0.1595E 03	-0.7294E 01	-0.7294E 01	0.1595E 03
5	SM2AL	0.0000	-0.4862E 01	-0.4862E 01	0.0000
6	SM2AT	0.3445E 03	0.1706E 04	0.1706E 04	0.3445E 03
6	SM2AT	0.0000	0.2324E 04	0.2324E 04	0.0000
7	SM2BL	-0.1662E 05	0.4632E 03	0.4632E 03	-0.1662E 05
7	SM2BL	0.0000	0.4314E 03	0.4314E 03	0.0000
8	SM2BT	0.7763E 03	0.3858E 04	0.3858E 04	0.7763E 03
8	SM2BT	0.0000	0.7882E 04	0.7882E 04	0.0000
9	SP2BL	-0.1662E 05	0.4632E 03	0.4632E 03	-0.1662E 05
9	SP2BL	0.0000	0.4314E 03	0.4314E 03	0.0000
10	SP2BT	0.7763E 03	0.3858E 04	0.3858E 04	0.7763E 03
10	SP2BT	0.0000	0.7882E 04	0.7882E 04	0.0000
11	SM3AL	0.1595E 03	-0.7294E 01	-0.7294E 01	0.1595E 03
11	SM3AL	0.0000	-0.4862E 01	-0.4862E 01	0.0000
12	SM3AT	-0.2126E 02	-0.9887E 02	-0.9887E 02	-0.2126E 02
12	SM3AT	0.0000	-0.6591E 02	-0.6591E 02	0.0000
13	SM3BL	-0.1662E 05	0.4632E 03	0.4632E 03	-0.1662E 05
13	SM3BL	0.0000	0.4314E 03	0.4314E 03	0.0000
14	SM3BT	0.2316E 03	0.1607E 04	0.1607E 04	0.2316E 03
14	SM3BT	0.0000	0.1497E 04	0.1497E 04	0.0000
15	SP3BL	-0.1662E 05	0.4632E 03	0.4632E 03	-0.1662E 05
15	SP3BL	0.0000	0.4314E 03	0.4314E 03	0.0000
16	SP3BT	0.2316E 03	0.1607E 04	0.1607E 04	0.2316E 03
16	SP3BT	0.0000	0.1497E 04	0.1497E 04	0.0000
17	SM12A	0.2137E-03	-0.2348E-02	-0.2348E-02	0.2137E-03
17	SM12A	0.0000	-0.2439E-02	-0.2439E-02	0.0000
18	SM12B	0.6592E-03	-0.6382E-02	-0.6382E-02	0.6592E-03
18	SM12B	0.0000	-0.9945E-02	-0.9945E-02	0.0000
19	SP12B	0.6592E-03	-0.6382E-02	-0.6382E-02	0.6592E-03
19	SP12B	0.0000	-0.9945E-02	-0.9945E-02	0.0000
20	SM13A	0.2137E-03	-0.2348E-02	-0.2348E-02	0.2137E-03
20	SM13A	0.0000	-0.2439E-02	-0.2439E-02	0.0000
21	SM13B	0.6592E-03	-0.6382E-02	-0.6382E-02	0.6592E-03
21	SM13B	0.0000	-0.9945E-02	-0.9945E-02	0.0000
22	SP13B	0.6592E-03	-0.6382E-02	-0.6382E-02	0.6592E-03
22	SP13B	0.0000	-0.9945E-02	-0.9945E-02	0.0000
23	SM23A	0.0000	0.0000	0.0000	0.0000
23	SM23A	0.0000	0.0000	0.0000	0.0000
24	SM23B	0.0000	0.0000	0.0000	0.0000
24	SM23B	0.0000	0.0000	0.0000	0.0000
25	SP23B	0.0000	0.0000	0.0000	0.0000
25	SP23B	0.0000	0.0000	0.0000	0.0000

NOTATION: S --- STRESS (SIGMA)  
 M --- MATRIX AND F --- FIBER  
 1,2,3 --- DIRECTIONS FOR STRESSES - PLY MATERIAL AXES  
 L,T --- DIRECTIONS OF PLY STRESSES  
 A --- REGION CONTAINING NO FIBERS  
 B --- REGION CONTAINING FIBERS AND MATRIX  
 EXAMPLE: SM2AL STANDS FOR TRANSVERSE NORMAL STRESS  
 IN REGION A DUE TO A LOAD IN THE LOGITUDINAL  
 DIRECTION

# Item 15(b)

## MICROSTRESS INFLUENCE COEFFICIENTS

THE FOLLOWING ARE THE MICROSTRESS INFLUENCE COEFFICIENTS FOR THE PRIMARY COMPOSITE AS--/INLS SYSTEM

INF. COEF.	SIGMA11 LBS/50.IN	SIGMA22 LBS/50.IN	SIGMA12 LBS/50.IN	SIGMA13 LBS/50.IN	SIGMA23 LBS/50.IN	DELTA T 1 DEG F	DELTA M IN
1 SM11	0.0290	0.4615	0.0000	0.0000	0.0000	-28.4291	-1079.0911
2 SM22A	0.0033	0.5103	0.0000	0.0000	0.0000	-10.1814	-1274.0061
3 SM22B	-0.3484	1.1678	0.0000	0.0000	0.0000	3.6376	443.8633
4 SM12A	0.0000	0.0000	0.3927	0.0000	0.0000	0.0000	0.0000
5 SM12B	0.0000	0.0000	1.2116	0.0000	0.0000	0.0000	0.0000
6 SM13A	0.0000	0.0000	0.0000	0.3927	0.0000	0.0000	0.0000
7 SM13B	0.0000	0.0000	0.0000	1.2116	0.0000	0.0000	0.0000
8 SM23A	0.0000	0.0000	0.0000	0.0000	0.9479	0.0000	0.0000
9 SM23B	0.0000	0.0000	0.0000	0.0000	1.4043	0.0000	0.0000
10 SM33A	0.0033	-0.0320	0.0000	0.0000	0.0000	-10.1814	-1274.0061
11 SM33B	-0.3484	0.3484	0.0000	0.0000	0.0000	3.6376	443.8633
12 SP11	1.7993	-0.3288	0.0000	0.0000	0.0000	21.4463	1944.1611
13 SP22B	-0.3484	1.1678	0.0000	0.0000	0.0000	3.6376	443.8633
14 SP33B	-0.3484	0.3484	0.0000	0.0000	0.0000	3.6376	443.8633
15 SP12	0.0000	0.0000	1.2116	0.0000	0.0000	0.0000	0.0000
16 SP13	0.0000	0.0000	0.0000	1.2116	0.0000	0.0000	0.0000
17 SP23B	0.0000	0.0000	0.0000	0.0000	1.4043	0.0000	0.0000

NOTE: TO OBTAIN THE ABSOLUTE VALUE OF THE MICROSTRESSES THE INF. COEF. SHOULD BE MULTIPLIED BY THE APPROPRIATE STRESSES OR THE TEMPERATURE GRADIENT OR THE MOISTURE CONTENT.

EXPLANATION: SM22B ,FOR EXAMPLE, STANDS FOR TRANSVERSE NORMAL STRESS INFLUENCE COEFFICIENT IN REGION B

## MICROSTRESS INFLUENCE COEFFICIENTS

THE FOLLOWING ARE THE MICROSTRESS INFLUENCE COEFFICIENTS FOR THE PRIMARY COMPOSITE SOLA/INHS SYSTEM

INF. COEF.	SIGMA11 LBS/50.IN	SIGMA22 LBS/50.IN	SIGMA12 LBS/50.IN	SIGMA13 LBS/50.IN	SIGMA23 LBS/50.IN	DELTA T 1 DEG F	DELTA M IN
1 SM11	0.0655	0.3323	0.0000	0.0000	0.0000	-28.7901	-2924.0504
2 SM22A	0.0059	0.3650	0.0000	0.0000	0.0000	-17.9027	-2187.4494
3 SM22B	-0.3484	0.8303	0.0000	0.0000	0.0000	0.2667	821.4043
4 SM12A	0.0000	0.0000	0.3643	0.0000	0.0000	0.0000	0.0000
5 SM12B	0.0000	0.0000	0.9902	0.0000	0.0000	0.0000	0.0000
6 SM13A	0.0000	0.0000	0.0000	0.3643	0.0000	0.0000	0.0000
7 SM13B	0.0000	0.0000	0.0000	0.9902	0.0000	0.0000	0.0000
8 SM23A	0.0000	0.0000	0.0000	0.0000	0.6091	0.0000	0.0000
9 SM23B	0.0000	0.0000	0.0000	0.0000	2.0423	0.0000	0.0000
10 SM33A	0.0059	-0.0214	0.0000	0.0000	0.0000	-17.9027	-2187.4494
11 SM33B	-0.3484	0.3484	0.0000	0.0000	0.0000	0.2667	821.4043
12 SP11	1.8037	-0.0886	0.0000	0.0000	0.0000	-14.8484	1222.4417
13 SP22B	-0.3484	0.8343	0.0000	0.0000	0.0000	0.2667	821.4043
14 SP33B	-0.3484	0.3484	0.0000	0.0000	0.0000	0.2667	821.4043
15 SP12	0.0000	0.0000	0.9902	0.0000	0.0000	0.0000	0.0000
16 SP13	0.0000	0.0000	0.0000	0.9902	0.0000	0.0000	0.0000
17 SP23B	0.0000	0.0000	0.0000	0.0000	2.0423	0.0000	0.0000

NOTE: TO OBTAIN THE ABSOLUTE VALUE OF THE MICROSTRESSES THE INF. COEF. SHOULD BE MULTIPLIED BY THE APPROPRIATE STRESSES OR THE TEMPERATURE GRADIENT OR THE MOISTURE CONTENT.

EXPLANATION: SM22B ,FOR EXAMPLE, STANDS FOR TRANSVERSE NORMAL STRESS INFLUENCE COEFFICIENT IN REGION B

## MICROSTRESS INFLUENCE COEFFICIENTS

THE FOLLOWING ARE THE MICROSTRESS INFLUENCE COEFFICIENTS FOR THE SECONDARY COMPOSITE AS--/INHS SYSTEM

INF. COEF.	SIGMA11 LBS/50.IN	SIGMA22 LBS/50.IN	SIGMA12 LBS/50.IN	SIGMA13 LBS/50.IN	SIGMA23 LBS/50.IN	DELTA T 1 DEG F	DELTA M IN
1 SM11	0.0437	0.3384	0.0000	0.0000	0.0000	-17.1007	-1950.7063
2 SM22A	0.0037	0.5034	0.0000	0.0000	0.0000	-9.9992	-1571.7661
3 SM22B	-0.3245	1.7004	0.0000	0.0000	0.0000	3.2438	510.8889
4 SM12A	0.0000	0.0000	0.3784	0.0000	0.0000	0.0000	0.0000
5 SM12B	0.0000	0.0000	1.5430	0.0000	0.0000	0.0000	0.0000
6 SM13A	0.0000	0.0000	0.3784	0.0000	0.0000	0.0000	0.0000
7 SM13B	0.0000	0.0000	1.5430	0.0000	0.0000	0.0000	0.0000
8 SM23A	0.0000	0.0000	0.0000	0.0000	0.4040	0.0000	0.0000
9 SM23B	0.0000	0.0000	0.0000	0.0000	1.0591	0.0000	0.0000
10 SM33A	0.0037	-0.0143	0.0000	0.0000	0.0000	-9.9992	-1571.7661
11 SM33B	-0.3245	0.3245	0.0000	0.0000	0.0000	3.2438	510.8889
12 SP11	2.7093	-0.3213	0.0000	0.0000	0.0000	66.7289	3050.1049
13 SP22B	-0.3245	1.7004	0.0000	0.0000	0.0000	3.2438	510.8889
14 SP33B	-0.3245	0.3245	0.0000	0.0000	0.0000	3.2438	510.8889
15 SP12	0.0000	0.0000	1.5430	0.0000	0.0000	0.0000	0.0000
16 SP13	0.0000	0.0000	0.0000	1.5430	0.0000	0.0000	0.0000
17 SP23B	0.0000	0.0000	0.0000	0.0000	1.0591	0.0000	0.0000

NOTE: TO OBTAIN THE ABSOLUTE VALUE OF THE MICROSTRESSES THE INF. COEF. SHOULD BE MULTIPLIED BY THE APPROPRIATE STRESSES OR THE TEMPERATURE GRADIENT OR THE MOISTURE CONTENT.

EXPLANATION: SM22B ,FOR EXAMPLE, STANDS FOR TRANSVERSE NORMAL STRESS INFLUENCE COEFFICIENT IN REGION B

# Item 16

## STRESS CONCENTRATION FACTORS (AROUND A CIRCULAR HOLE)

NOTE: K<sub>1XX</sub> --> STRESS CONCENTRATION FACTOR DUE TO SIGMA XX  
 K<sub>1YY</sub> --> STRESS CONCENTRATION FACTOR DUE TO SIGMA YY  
 K<sub>1XY</sub> --> STRESS CONCENTRATION FACTOR DUE TO SIGMA XY  
 LAYUP --> 0 90 90 0

THETA	K <sub>1XX</sub>	K <sub>1YY</sub>	K <sub>1XY</sub>	THETA	K <sub>1XX</sub>	K <sub>1YY</sub>	K <sub>1XY</sub>
0.0	-0.6160	3.8562	0.0000	180.0	-0.6160	3.8562	0.0000
5.0	-0.5709	3.6729	-1.1975	185.0	-0.5709	3.6729	-1.1975
10.0	-0.4972	3.2156	-2.1209	190.0	-0.4972	3.2156	-2.1209
15.0	-0.3168	2.6649	-2.6886	195.0	-0.3168	2.6649	-2.6886
20.0	-0.1799	2.1516	-2.9777	200.0	-0.1799	2.1516	-2.9777
25.0	-0.0569	1.7253	-3.1020	205.0	-0.0569	1.7253	-3.1020
30.0	0.0532	1.3675	-3.1493	210.0	0.0532	1.3675	-3.1493
35.0	0.1566	1.1225	-3.1741	215.0	0.1566	1.1225	-3.1741
40.0	0.2613	0.9117	-3.2000	220.0	0.2613	0.9117	-3.2000
45.0	0.3764	0.7383	-3.2701	225.0	0.3764	0.7383	-3.2701
50.0	0.5138	0.5879	-3.3730	230.0	0.5138	0.5879	-3.3730
55.0	0.6899	0.4472	-3.5263	235.0	0.6899	0.4472	-3.5263
60.0	0.9302	0.3018	-3.7367	240.0	0.9302	0.3018	-3.7367
65.0	1.2764	0.1530	-4.0029	245.0	1.2764	0.1530	-4.0029
70.0	1.7980	-0.0865	-4.2958	250.0	1.7980	-0.0865	-4.2958
75.0	2.6023	-0.3969	-4.5007	255.0	2.6023	-0.3969	-4.5007
80.0	3.7983	-0.8379	-4.2743	260.0	3.7983	-0.8379	-4.2743
85.0	5.2286	-1.3549	-2.9009	265.0	5.2286	-1.3549	-2.9009
90.0	5.9765	-1.6233	-0.0001	270.0	5.9765	-1.6233	-0.0001
95.0	5.2287	-1.3549	2.9007	275.0	5.2287	-1.3549	2.9007
100.0	3.7984	-0.8379	4.2743	280.0	3.7984	-0.8379	4.2743
105.0	2.6026	-0.3969	4.5007	285.0	2.6026	-0.3969	4.5007
110.0	1.7980	-0.0865	4.2958	290.0	1.7980	-0.0865	4.2958
115.0	1.2764	0.1530	4.0029	295.0	1.2764	0.1530	4.0029
120.0	0.9302	0.3018	3.7367	300.0	0.9302	0.3018	3.7367
125.0	0.6899	0.4472	3.5263	305.0	0.6899	0.4472	3.5263
130.0	0.5138	0.5879	3.3730	310.0	0.5138	0.5879	3.3730
135.0	0.3764	0.7383	3.2702	315.0	0.3764	0.7383	3.2702
140.0	0.2613	0.9116	3.2001	320.0	0.2613	0.9116	3.2001
145.0	0.1567	1.1225	3.1741	325.0	0.1567	1.1225	3.1741
150.0	0.0532	1.3675	3.1493	330.0	0.0532	1.3675	3.1493
155.0	-0.0569	1.7252	3.1021	335.0	-0.0569	1.7252	3.1021
160.0	-0.1799	2.1515	2.9777	340.0	-0.1799	2.1515	2.9777
165.0	-0.3168	2.6648	2.6886	345.0	-0.3168	2.6648	2.6886
170.0	-0.4971	3.2155	2.1210	350.0	-0.4971	3.2155	2.1211
175.0	-0.5709	3.6728	1.1976	355.0	-0.5709	3.6728	1.1976

# Item 17

## LOCATIONS OF PROBABLE DELAMINATION

RESULTS FOR PLY NO. 1 ORIENTATION 0.0 MATERIAL AS--IMLS AS--IMLS

CRITERION	RANGE	VALUE	LOCATION #
MAX OF K1XX*(NUCRT-MULRT)	0.0-- 90.0	0.588	75.0
MAX OF K1YY*(NUCRT-MULRT)	0.0-- 90.0	0.822	0.0
MAX OF K1XY*(NUCRT-MULRT)	0.0-- 90.0	1.411	60.0
MAX OF K1XX*(NUCRT-MULRT)	90.0-- 180.0	0.588	105.0
MAX OF K1YY*(NUCRT-MULRT)	90.0-- 180.0	0.822	180.0
MAX OF K1XY*(NUCRT-MULRT)	90.0-- 180.0	1.411	120.0
MAX OF K1XX*(NUCRT-MULRT)	180.0-- 270.0	0.588	235.0
MAX OF K1YY*(NUCRT-MULRT)	180.0-- 270.0	0.822	180.0
MAX OF K1XY*(NUCRT-MULRT)	180.0-- 270.0	1.411	240.0
MAX OF K1XX*(NUCRT-MULRT)	270.0-- 0.0	0.588	285.0
MAX OF K1YY*(NUCRT-MULRT)	270.0-- 0.0	0.822	0.0
MAX OF K1XY*(NUCRT-MULRT)	270.0-- 0.0	1.411	300.0

RESULTS FOR PLY NO. 2 ORIENTATION 90.0 MATERIAL SOLAWMS AS--IPMS

CRITERION	RANGE	VALUE	LOCATION #
MAX OF K1XX*(NUCRT-MULRT)	0.0-- 90.0	1.407	90.0
MAX OF K1YY*(NUCRT-MULRT)	0.0-- 90.0	1.087	15.0
MAX OF K1XY*(NUCRT-MULRT)	0.0-- 90.0	1.476	25.0
MAX OF K1XX*(NUCRT-MULRT)	90.0-- 180.0	1.407	90.0
MAX OF K1YY*(NUCRT-MULRT)	90.0-- 180.0	1.087	165.0
MAX OF K1XY*(NUCRT-MULRT)	90.0-- 180.0	1.476	155.0
MAX OF K1XX*(NUCRT-MULRT)	180.0-- 270.0	1.407	270.0
MAX OF K1YY*(NUCRT-MULRT)	180.0-- 270.0	1.087	195.0
MAX OF K1XY*(NUCRT-MULRT)	180.0-- 270.0	1.476	205.0
MAX OF K1XX*(NUCRT-MULRT)	270.0-- 0.0	1.407	270.0
MAX OF K1YY*(NUCRT-MULRT)	270.0-- 0.0	1.087	345.0
MAX OF K1XY*(NUCRT-MULRT)	270.0-- 0.0	1.476	335.0

RESULTS FOR PLY NO. 3 ORIENTATION 90.0 MATERIAL SOLAWMS AS--IPMS

CRITERION	RANGE	VALUE	LOCATION #
MAX OF K1XX*(NUCRT-MULRT)	0.0-- 90.0	1.407	90.0
MAX OF K1YY*(NUCRT-MULRT)	0.0-- 90.0	1.087	15.0
MAX OF K1XY*(NUCRT-MULRT)	0.0-- 90.0	1.476	25.0
MAX OF K1XX*(NUCRT-MULRT)	90.0-- 180.0	1.407	90.0
MAX OF K1YY*(NUCRT-MULRT)	90.0-- 180.0	1.087	165.0
MAX OF K1XY*(NUCRT-MULRT)	90.0-- 180.0	1.476	155.0
MAX OF K1XX*(NUCRT-MULRT)	180.0-- 270.0	1.407	270.0
MAX OF K1YY*(NUCRT-MULRT)	180.0-- 270.0	1.087	195.0
MAX OF K1XY*(NUCRT-MULRT)	180.0-- 270.0	1.476	205.0
MAX OF K1XX*(NUCRT-MULRT)	270.0-- 0.0	1.407	270.0
MAX OF K1YY*(NUCRT-MULRT)	270.0-- 0.0	1.087	345.0
MAX OF K1XY*(NUCRT-MULRT)	270.0-- 0.0	1.476	335.0

RESULTS FOR PLY NO. 4 ORIENTATION 0.0 MATERIAL AS--IMLS AS--IMLS

CRITERION	RANGE	VALUE	LOCATION #
MAX OF K1XX*(NUCRT-MULRT)	0.0-- 90.0	0.588	75.0
MAX OF K1YY*(NUCRT-MULRT)	0.0-- 90.0	0.822	0.0
MAX OF K1XY*(NUCRT-MULRT)	0.0-- 90.0	1.411	60.0
MAX OF K1XX*(NUCRT-MULRT)	90.0-- 180.0	0.588	105.0
MAX OF K1YY*(NUCRT-MULRT)	90.0-- 180.0	0.822	180.0
MAX OF K1XY*(NUCRT-MULRT)	90.0-- 180.0	1.411	120.0
MAX OF K1XX*(NUCRT-MULRT)	180.0-- 270.0	0.588	235.0
MAX OF K1YY*(NUCRT-MULRT)	180.0-- 270.0	0.822	180.0
MAX OF K1XY*(NUCRT-MULRT)	180.0-- 270.0	1.411	240.0
MAX OF K1XX*(NUCRT-MULRT)	270.0-- 0.0	0.588	285.0
MAX OF K1YY*(NUCRT-MULRT)	270.0-- 0.0	0.822	0.0
MAX OF K1XY*(NUCRT-MULRT)	270.0-- 0.0	1.411	300.0

NOTES: K1XX --> STRESS CONCENTRATION FACTOR DUE TO SIGMA XX  
 K1YY --> STRESS CONCENTRATION FACTOR DUE TO SIGMA YY  
 K1XY --> STRESS CONCENTRATION FACTOR DUE TO SIGMA XY  
 MULRT --> PLY POISSON RATIO IN R AND T AXES  
 NUCRT --> COMPOSITE POISSON RATIO IN R AND T AXES  
 ( R AND T ARE THE RADIAL AND THE TANGENTIAL DIRECTIONS)  
 ONLY 5 DEG. INTERVALS ARE CONSIDERED. THE ACTUAL VALUE  
 IS EXPECTED TO BE WITHIN 5 DEG. OF THE PRINTED RESULT.

## Item 18

## PLY STRESS AND STRAIN INFLUENCE COEFFICIENTS ARRAYS

PLY NO.	MATERIAL SYSTEM	THETA	RESPONSE	NX (UNIT LOAD ...)	NY LB./INCH	NXY	MX (UNIT MOMENT ...)	MY LB.IN/INCH	MXY	DELTA T (1 DEG F)	DELTA M (1 %)
1	AS--/IMLS	0.0	EPS11	2.7511	-0.2236	0.0000	266.0017	-59.2159	0.0000	1.1801	109.0192
			EPS22	-0.2236	7.2490	-0.0000	-59.2160	2938.3239	-0.0024	3.0164	342.4929
			EPS12	0.0000	-0.0000	54.0002	0.0000	-0.0024	8011.9614	-0.0000	-0.0023
			SIG11	47.6932	-1.4628	0.0000	4590.0906	-47.6819	-0.0001	11.7478	588.3518
			SIG22	0.6647	8.1391	-0.0001	21.6664	3309.6074	-0.0027	-20.9790	-1238.7244
			SIG12	0.0000	-0.0000	29.5831	0.0000	-0.0013	4382.7227	-0.0000	-0.0012
2	SGLA/MPMS	90.0	AS--/IMMS								
			EPS11	-0.2236	7.2490	0.0000	-14.0040	734.5819	0.0019	3.0164	342.4929
			EPS22	2.7511	-0.2236	-0.0001	66.5005	-14.0040	-0.0023	1.1801	109.0192
			EPS12	-0.0000	0.0001	-54.0002	-0.0002	0.0023	-2002.9873	0.0001	0.0029
			SIG11	-1.3294	83.7219	0.0002	-140.8320	8487.6133	0.0213	61.9561	2677.6478
			SIG22	4.6136	2.9237	-0.0001	187.2342	389.9329	-0.0034	-23.4957	-1176.7069
			SIG12	-0.0000	0.0001	-40.8337	-0.0002	0.0019	-1512.3738	0.0000	0.0022
3	SGLA/MPMS	90.0	AS--/IMMS								
			EPS11	-0.2236	7.2490	0.0000	14.0040	-734.5796	-0.0019	3.0164	342.4927
			EPS22	2.7511	-0.2236	-0.0001	-66.5004	14.0040	0.0023	1.1801	109.0192
			EPS12	-0.0000	0.0001	-54.0002	0.0002	0.0023	2002.9846	0.0001	0.0029
			SIG11	-1.3294	83.7219	0.0002	-140.8316	-8487.5896	-0.0213	61.9561	2677.6451
			SIG22	4.6136	2.9237	-0.0001	187.2300	-389.9319	0.0034	-23.4957	-1176.7069
			SIG12	-0.0000	0.0001	-40.8337	0.0002	-0.0019	1512.3713	0.0000	0.0022
4	AS--/IMLS	0.0	EPS11	2.7511	-0.2236	0.0000	-266.0017	59.2159	-0.0000	1.1801	109.0192
			EPS22	-0.2236	7.2490	-0.0000	59.2159	-2938.3228	0.0024	3.0164	342.4926
			EPS12	0.0000	-0.0000	54.0002	-0.0000	0.0024	-8011.9614	-0.0000	-0.0023
			SIG11	47.6932	-1.4628	0.0000	-4590.0906	47.6819	0.0001	11.7478	588.3518
			SIG22	0.6647	8.1391	-0.0001	-21.6664	-3309.6067	0.0027	-20.9790	-1238.7231
			SIG12	0.0000	-0.0000	29.5831	-0.0000	0.0013	-4382.7227	-0.0000	-0.0012

NOTE: STRAINS ARE IN MICRO INCH/INCH.  
STRESSES ARE IN POUNDS/INCH SQ.

## EXPLANATION OF THE INFLUENCE COEFFICIENTS

NX, NY AND NXY ARE UNIT LOADS IN LB./INCH. MX, MY AND MXY ARE UNIT MOMENTS IN LB.IN/INCH. DELTA T IS A UNIT TEMP. DIFF. AND DELTA M IS A UNIT PERCENTAGE OF MOISTURE CONTENT. TO OBTAIN RESPONSE R FOR A GENERAL APPLIED LOAD VECTOR P USE THE FOLLOWING EQUATION:

$$(R) = (AIMP) \times (P)$$

NOTE: R IS A 6X1 COLUMN VECTOR DEFINED BY

$$(R) = (EPS11 \ EPS22 \ EPS12 \ SIG11 \ SIG12 \ SIG12)^T$$

P IS A 8X1 COLUMN VECTOR DEFINED BY

$$(P) = (NX \ NY \ NXY \ MX \ MY \ MXY \ DELTA T \ DELTA M)^T$$

AIMP IS A (6X8) MATRIX CONTAINING THE INFLUENCE COEFFICIENTS ARRAYS.

## PLY STRESS INFLUENCE COEFFICIENTS ARRAYS

PLY NO.	MATERIAL SYSTEM	THETA	RESPONSE	NX (UNIT LOAD ... SCALE FACTOR = (1/TC)	NY LB./INCH 33.333	NXY LB./INCH	MX (UNIT MOMENT ... SCALE FACTOR = 6666.668 (6/TCX02)	MY LB.IN/INCH	MXY	DELTA T (1 DEG F) 13.394	DELTA M (1 %) 1223.138	
1	AS--/IMLS	0.0	SIG11	1.4308	-0.0439	0.0000	0.6898	-0.0072	-0.0000	0.8797	0.4810	
			SIG22	0.0199	0.2442	-0.0000	0.0032	0.4964	-0.0000	-1.5709	-1.0127	
			SIG12	0.0000	-0.0000	0.6879	0.0000	-0.0000	0.6374	-0.0000	-0.0000	
2	SGLA/MPMS	90.0	AS--/IMMS	SIG11	-0.0399	2.5117	0.0000	-0.0211	1.2731	0.0000	3.1619	2.0255
			SIG22	0.1384	0.0878	-0.0000	0.0161	0.0469	-0.0000	-1.7594	-0.9620	
			SIG12	-0.0000	0.0000	-1.2250	-0.0000	0.0000	-0.2269	0.0000	0.0000	
3	SGLA/MPMS	90.0	AS--/IMMS	SIG11	-0.0399	2.5117	0.0000	0.0211	-1.2731	-0.0000	3.1619	2.0255
			SIG22	0.1384	0.0878	-0.0000	-0.0161	-0.0469	0.0000	-1.7594	-0.9620	
			SIG12	-0.0000	0.0000	-1.2250	0.0000	-0.0000	0.2269	0.0000	0.0000	
4	AS--/IMLS	0.0	SIG11	1.4308	-0.0439	0.0000	-0.6898	0.0072	0.0000	0.8797	0.4810	
			SIG22	0.0199	0.2442	-0.0000	-0.0032	-0.4964	0.0000	-1.5709	-1.0127	
			SIG12	0.0000	-0.0000	0.6879	-0.0000	0.0000	-0.6374	-0.0000	-0.0000	

NOTE: THE MEMBRANE STRESSES ARE NORMALIZED W.R.T THE AVERAGE STRESS DUE TO UNIT LOAD IN AN EQUIVALENT HOMOGENEOUS SECTION. THE BENDING STRESSES ARE NORMALIZED W.R.T THE MAXIMUM STRESS DUE TO UNIT MOMENT. THE TEMPERATURE AND MOISTURE STRESSES ARE NORMALIZED W.R.T THE AVERAGE STRESSES DUE TO UNIT TEMPERATURE DIFFERENCE AND UNIT PERCENTAGE OF MOISTURE. TO OBTAIN THE ABSOLUTE VALUES OF THE STRESSES THE INFLUENCE COEFFICIENTS SHOULD BE MULTIPLIED BY THE INDICATED SCALE FACTORS. THESE SHOULD BE MULTIPLIED BY THE CORRESPONDING LOADS TO OBTAIN STRESSES IN THE PLYS.

## Item 19

## LAMINATE FAILURE STRESS ANALYSIS

LAYUP --> 0 90 90 0							
LAMINATE FAILURE STRESSES BASED UPON FIRST PLY FAILURE CRITERIA (NO TEMPERATURE OR MOISTURE STRESSES)							
PLY NO.	1	THETA	0.00	MATERIAL SYSTEM = AS--IMLS AS--IMLS			
LOADS	SL11T	SL11C	SL22T	SL22C	SL12S	FAIL. LOAD	MODE
	222.7741	87.6392	9.8865	15.8194	9.1261	KSI	
	KSI	KSI	KSI	KSI	KSI	KSI	
SCXXT MIN (	155.699	-61.252	251.863	-753.138	0.0000	155.699	SL11T
SCXXC MIN (	-155.699	61.252	-251.863	753.138	0.0000	61.252	SL11C
SCYXT MIN (	-5076.389	1997.815	20.504	-61.511	0.0000	20.504	SL22T
SCYXC MIN (	5076.389	-1997.815	-20.504	61.511	0.0000	61.511	SL22C
SCYYS MIN (	0.0000	0.0000	*****	*****	9.776	9.776	SL12S

LAMINATE FAILURE STRESSES BASED UPON FIRST PLY FAILURE CRITERIA (NO TEMPERATURE OR MOISTURE STRESSES)							
PLY NO.	2	THETA	90.00	MATERIAL SYSTEM = SGLAMHS AS--IMLS			
LOADS	SL11T	SL11C	SL22T	SL22C	SL12S	FAIL. LOAD	MODE
	216.8321	166.5112	9.9151	23.1353	11.9513	KSI	
	KSI	KSI	KSI	KSI	KSI	KSI	
SCXXT MIN (	-5436.881	4175.863	71.638	-167.155	0.0000	71.638	SL22T
SCXXC MIN (	5436.881	-4175.863	-71.638	167.155	0.0000	167.155	SL22C
SCYXT MIN (	86.330	-66.295	112.967	-263.589	*****	86.330	SL11T
SCYXC MIN (	-86.330	66.295	-112.967	263.589	*****	66.295	SL11C
SCYYS MIN (	*****	*****	*****	*****	9.756	9.756	SL12S

LAMINATE FAILURE STRESSES BASED UPON FIRST PLY FAILURE CRITERIA (NO TEMPERATURE OR MOISTURE STRESSES)							
PLY NO.	3	THETA	90.00	MATERIAL SYSTEM = SGLAMHS AS--IMLS			
LOADS	SL11T	SL11C	SL22T	SL22C	SL12S	FAIL. LOAD	MODE
	216.8321	166.5112	9.9151	23.1353	11.9513	KSI	
	KSI	KSI	KSI	KSI	KSI	KSI	
SCXXT MIN (	-5436.881	4175.863	71.638	-167.155	0.0000	71.638	SL22T
SCXXC MIN (	5436.881	-4175.863	-71.638	167.155	0.0000	167.155	SL22C
SCYXT MIN (	86.330	-66.295	112.967	-263.589	*****	86.330	SL11T
SCYXC MIN (	-86.330	66.295	-112.967	263.589	*****	66.295	SL11C
SCYYS MIN (	*****	*****	*****	*****	9.756	9.756	SL12S

NOTE: "\*\*\*\*\*" IMPLIES -"NOT APPLICABLE"-

LAYUP --> 0 90 90 0							
LAMINATE FAILURE STRESSES BASED UPON FIRST PLY FAILURE CRITERIA (NO TEMPERATURE OR MOISTURE STRESSES)							
PLY NO.	4	THETA	0.00	MATERIAL SYSTEM = AS--IMLS AS--IMLS			
LOADS	SL11T	SL11C	SL22T	SL22C	SL12S	FAIL. LOAD	MODE
	222.7741	87.6392	9.8865	15.8194	9.1261	KSI	
	KSI	KSI	KSI	KSI	KSI	KSI	
SCXXT MIN (	155.699	-61.252	251.862	-753.187	0.0000	155.699	SL11T
SCXXC MIN (	-155.699	61.252	-251.862	753.187	0.0000	61.252	SL11C
SCYXT MIN (	-5076.389	1997.817	20.504	-61.511	0.0000	20.504	SL22T
SCYXC MIN (	5076.389	-1997.817	-20.504	61.511	0.0000	61.511	SL22C
SCYYS MIN (	0.0000	0.0000	*****	*****	9.776	9.776	SL12S

**Item 20**  
**S U M M A R Y**

**LAMINATE FAILURE STRESS ANALYSIS - (NO TEMPERATURE OR MOISTURE STRESSES)**  
**(BASED UPON FIRST PLY FAILURE)**

LOAD TYPE	STRESS KSI	FAILURE MODE	PLY NO.	THETA	MATERIAL SYSTEM	
SCXKT	71.638	SL22T	3	90.0	SOLANWHS	AS--TWHS
SCXKC	61.252	SL11C	4	0.0	AS--TWLS	AS--TWLS
SCYTT	28.504	SL22T	1	0.0	AS--TWLS	AS--TWLS
SCYYC	61.511	SL22C	1	0.0	AS--TWLS	AS--TWLS
SCXYS	5.776	SL12S	4	0.0	AS--TWLS	AS--TWLS

**LAMINATE FAILURE STRESS ANALYSIS - (NO TEMPERATURE OR MOISTURE STRESSES)**  
**(BASED UPON FIBER FAILURE)**

LOAD TYPE	STRESS KSI	FAILURE MODE	PLY NO.	THETA	MATERIAL SYSTEM	
SCXKT	155.699	SL11T	4	0.0	AS--TWLS	AS--TWLS
SCXKC	61.252	SL11C	4	0.0	AS--TWLS	AS--TWLS
SCYTT	86.330	SL11T	2	90.0	SOLANWHS	AS--TWHS
SCYYC	66.295	SL11C	2	90.0	SOLANWHS	AS--TWHS
SCXYS	*****	N/A				

NOTE: IF THERE IS NO ANGLE PLY "SCXYS" BASED UPON FIBER FAILURE IS NOT PREDICTED.

## Appendix C

### Resident Data Bank (FBMTDATA.BANK)

T300

```

FP 3000 0.300E-03 0.640E-01
FE 0.320E 08 0.200E 07 0.200E 00 0.250E 00 0.130E 07 0.700E 06
FT -0.550E-06 0.560E-05 0.580E 03 0.580E 02 0.170E 00
FS 0.350E 06 0.300E 06 0.000 0.000 0.000 0.000

```

AS--

```

FP 10000 0.300E-03 0.630E-01
FE 0.310E 08 0.200E 07 0.200E 00 0.250E 00 0.200E 07 0.100E 07
FT -0.550E-06 0.560E-05 0.580E 03 0.580E 02 0.170E 00
FS 0.400E 06 0.400E 06 0.000 0.000 0.000 0.000

```

SGLA

```

FP 204 0.360E-03 0.900E-01
FE 0.124E 08 0.124E 08 0.200E 00 0.200E 00 0.517E 07 0.517E 07
FT 0.280E-05 0.280E-05 0.750E 01 0.750E 01 0.170E 00
FS 0.360E 06 0.300E 06 0.360E 06 0.300E 06 0.180E 06 0.180E 06

```

HMSF HIGH MODULUS SURFACE TREATED FIBER.

```

FP 10000 0.300E-03 0.703E-01
FE 0.550E 08 0.900E 06 0.200E 00 0.250E 00 0.110E 07 0.700E 06
FT -0.550E-06 0.560E-05 0.580E 03 0.580E 02 0.170E 00
FS 0.280E 06 0.200E 06 0.000 0.000 0.000 0.000

```

OVER END OF FIBER PROPERTIES.

IMLS INTERMEDIATE MODULUS LOW STRENGTH MATRIX.

```

MP 0.460E-01
ME 0.500E 06 0.410E 00 0.570E-04
MT 0.125E 01 0.250E 00
MS 0.700E 04 0.210E 05 0.700E 04 0.140E-01 0.420E-01 0.320E-01 0.320E-01
MV 0.225E 00 0.420E 03

```

IMHS INTERMEDIATE MODULUS HIGH STRENGTH MATRIX.

```

MP 0.440E-01
ME 0.500E 06 0.350E 00 0.360E-04
MT 0.125E 01 0.250E 00
MS 0.150E 05 0.350E 05 0.130E 05 0.200E-01 0.500E-01 0.350E-01 0.350E-01
MV 0.225E 00 0.420E 03

```

HMS HIGH MODULUS HIGH STRENGTH MATRIX.

```

MP 0.450E-01
ME 0.750E 06 0.350E 00 0.400E-04
MT 0.125E 01 0.250E 00
MS 0.200E 05 0.500E 05 0.150E 05 0.200E-01 0.500E-01 0.400E-01 0.400E-01
MV 0.225E 00 0.420E 03

```

OVER END OF MATRIX PROPERTIES.



## References

1. Chamis, Christos C.: Design Oriented Analysis and Synthesis of Multilayered-Filamentary Structural Panels. Ph.D. Thesis, Case Western Reserve Univ., 1967.
2. Chamis, Christos C.: Thermoelastic Properties of Unidirectional Filamentary Composites by a Semiempirical Micromechanics Theory. Advanced Techniques for Material Investigation and Fabrication. Science of Advanced Materials and Process Engineering Proceedings. Vol. 14. Western Periodicals Co., 1968, paper 1-4-5.
3. Chamis, Christos C.: Characterization and Design Mechanics for Fiber-Reinforced Metals. NASA TN-D-5784, 1970.
4. Chamis, Christos C.: Failure Criteria for Filamentary Composites: NASA TN-D-5367, 1969.
5. Chamis, Christos C.: Buckling of Anisotropic Composite Plates. J. Struct. Div., Am. Soc. Civ. Eng., vol. 95, no. ST10, Oct. 1969, pp. 2119-2139.
6. Chamis, Christos C.: Design and Analysis of Fiber Composite Structural Components. Aerospace Structural Materials. NASA SP-227, 1970, pp. 217-228.
7. Foye, R.L.: Compression Strength of Unidirectional Composites. AIAA Paper 66-143, Jan. 1966.
8. Ozisik, M.N.: Boundary Value Problems of Heat Conduction. International Textbook Co., 1968.
9. Hoffman, Oscar: The Brittle Strength of Orthotropic Materials. J. Compos. Mater., vol. 1, no. 2, Apr. 1967, pp. 200-206.
10. Chamis, C.C.: Important Factors in Fiber Composite Design. Proceedings, Society of the Plastics Industry, Annual Technical Conference, 24th, 1969, pp. 18-E-1 to 18-E-13.
11. Chamis, C.C.: Computer Code for the Analysis of Multilayered Fiber Composites - User's Manual. NASA TN-D-7013, 1971.
12. Chamis, C.C., and Sinclair, J.H.: Prediction of Properties of Intraply Hybrid Composites. NASA TM-79087, 1979.
13. Chamis, C.C., and Sinclair, J.H.: Micromechanics of Intraply Hybrid Composites: Elastic and Thermal Properties. NASA TM-79253, 1979.
14. Chamis, C.C., Lark, R.F., and Sinclair, J.H.: Mechanical Property Characterization of Intraply Hybrid Composites. NASA TM-79306, 1979.
15. Chamis, C.C., Lark, R.F., and Sinclair, J.H.: An Integrated Theory for Predicting the Hydrothermomechanical Response of Advanced Composite Structural Components. NASA TM-73812, 1977.
16. Chamis, C.C., and Sinclair, J.H.: Computer Code for Intraply Hybrid Composite Design. NASA TM-82593, 1981.
17. Chamis, C.C., and Sinclair, J.H.: INHYD: Computer Code for Intraply Hybrid Composite Design. NASA TP-2239, 1983.
18. Pagano, N.J., Pipes, R.B.: Some Observations on the Interlaminar Strength of Composite Laminates. Int. J. Mech. Sci., vol. 15, no. 8, 1973 pp. 679-688.
19. Lekhnitskii, S.G. (P. Fern, Transl.): Theory of Elasticity of an Anisotropic Elastic Body. Holden-Day, 1963.



**APPENDIX 4.**  
**PATRAM PREFERENCE MANUAL**



## **CSTEM PREFERENCE FOR PATRAN**



# TABLE OF CONTENTS

<b>SCOPE .....</b>	<b>1</b>
<b>INSTALLATION .....</b>	<b>1</b>
<b>CSTEM PREFERENCE .....</b>	<b>2</b>
CSTEM INPUT DECK GENERATION .....	2
GEOMETRY .....	3
MESH GENERATION .....	3
MATERIAL PROPERTIES .....	4
Isotropic .....	4
Orthotropic .....	5
ELEMENT PROPERTIES .....	6
Homogeneous Elements .....	6
Layered Elements .....	6
Unspecified Element Properties .....	9
LAYERING DEFINITION .....	11
Step 1: .....	11
Step 2: .....	11
Step 3: .....	14
Step 4: .....	16
Layering Recommendations .....	17
LOAD CASES .....	18
LOADS AND BCS .....	19
ANALYSIS .....	20
Action .....	20
Object .....	20
Method .....	21
Write Input Deck .....	21
Analysis Controls .....	21
Load Case Controls .....	21
Specify Generation Sets .....	21
Select Load Cases .....	23
Read Layer Gen Sets .....	24
Select Layering Fields .....	25
Layer Orientation Visualization .....	26
Results .....	27
Translate .....	27
Template .....	27
Create XYDATA Files .....	29
<b>P3 DATABASE TO SIESTA RDB TRANSLATOR .....</b>	<b>30</b>
FRAMES .....	30
NODES .....	31
ELEMS .....	31
MPCS .....	32
MATL .....	32
BCS .....	35
BSURF .....	37
CONTROL VARIABLES .....	38
HEAT TRANSFER INPUT DECK .....	40
STRUCTURAL INPUT DECK .....	41
CSTEM DECK GENERATOR .....	41





## SCOPE

This document describes the CSTEM preference for PATRAN, which can be used to generate a CSTEM input deck and to process CSTEM analysis results. The preference includes the specification of CSTEM related parameters for PATRAN, a translator from the PATRAN database to a SIESTA database, and a CSTEM deck generator which operates primarily on the SIESTA database. Users of the CSTEM preference should be able to generate a CSTEM input deck from scratch from within PATRAN. Most CSTEM capabilities are accessible from the preference, although some lesser utilized capabilities can only be accessed by direct manipulation of the CSTEM input deck.

This document addresses topics in two parts that may be described as a User Manual (CSTEM PREFERENCE) and a Programmers Manual (P3 DATABASE TO SIESTA RDB TRANSLATOR). Users familiar with PATRAN3 should have little difficulty utilizing the preference with the possible exception of the Analysis form, which includes specification of CSTEM control variables and element layering.

## INSTALLATION

The preference is contained in a number of files which are either PCL or FORTRAN source files. The PCL files are compiled into PATRAN libraries. FORTRAN source files are compiled into SIESTA libraries or into the P3 to SIESTA database translator (`pdb2deck.x`), the CSTEM input deck generator (`RDB2CSD`), or the results file translator (`results_translator`). An awk script, `awk.post4`, is also included as part of the preference.

A script called `make_pdb2deck` is used to create the P3 to SIESTA database translator. The script requires modification to specify the location of the PATRAN libraries, compile options, and paths to source files, SIESTA libraries, and executables at a specific installation site. A makefile called `makedeck` is used to compile the deck generator. The makefile may require modification to specify the location of the source files and SIESTA libraries. A script called `make_results` is used to create the results translator. The script requires modification to specify the location of the results translator source files and executable. These three compilation files are used to generate the executables and are not required at each utilization of the preference.

A `p3epilog.pcl` file, which is executed whenever PATRAN is entered, contains the PATRAN commands to compile the CSTEM preference into the `cstem.plb` library. This library may be created and made available in such a way that it is readily available at a specific site so that the compilation is not necessary at each utilization of the preference. The `p3epilog.pcl` file also contains paths to the P3 to SIESTA database translator executable, `pdb2deck.x`, the deck generator executable, `RDB2CSD`, and the results translator executable `results_translator`. These paths must always be present in the `p3epilog.pcl` file to utilize the preference.

The P3 to SIESTA database translator is called by the CSTEM preference from within PATRAN. This is done with a script, `wrdeck`, written from the preference. The script contains locations of executables as defined in the `p3epilog.pcl` file.

The CSTEM deck generator is called by the P3 to SIESTA database translator. The site specific location of the deck generator executable, `RDB2CSD`, is defined by the environment variable `EWSP` as set in the `wrdeck` script. This `wrdeck` environment variable is defined to be the same as the variable `deck_path` in the `p3epilog.pcl` file.

## CSTEM PREFERENCE

The CSTEM preference customizes PATRAN for use with the CSTEM analysis code by setting various PATRAN database quantities and properties specific to the CSTEM application. Most PATRAN forms and menus maintain a common appearance as with any other analysis code, but only options pertaining to the CSTEM analysis code will be available.

Compiling the CSTEM preference into the appropriate PATRAN libraries is generally a one time operation performed by computer systems personnel at the user's site. Once this has been done, the CSTEM preference is loaded into the currently open database by issuing the command "load\_cstem()" in the PATRAN command window. When the CSTEM preference is loaded, it first loads the generic PATRAN definitions then activates and utilizes the generic definitions as much as possible, creating new definitions specific to CSTEM if necessary. The user may create a database template loaded with the CSTEM preference in the usual manner.

Use of the preference to generate CSTEM input decks results in a CSTEM structural input deck written to file f35.dat and a CSTEM heat transfer input deck written to file f34.dat. Processing of CSTEM analysis results is also available from the preference.

Changing to a new version of PATRAN will require that the cstem.plb library be recreated by compiling the CSTEM preference files using the new version of PATRAN. The fortran executables, pdb2deck.x and RDB2CSD, will also need to be recompiled. Any necessary modifications to the compile script, make\_pdb2deck, will usually be indicated in the link\_fort\_access file provided with the new PATRAN version in the customization subdirectory.

The version of the preference can be determined by issuing the command "cstem\_pref\_version()" in the PATRAN command window.

## CSTEM INPUT DECK GENERATION

The resultant of using the CSTEM preference is the generation of structural and heat transfer input decks for the CSTEM finite element code. This process is activated by using the Analysis form, which is more fully described in a later section. The method used to generate these input decks is to utilize the deck generator contained in the SIESTA program, so that the PATRAN database is translated into a SIESTA database as the first step of generating the CSTEM input decks. Certain functions of the deck generation process are performed external to SIESTA, since they utilize data that can not be directly translated into the SIESTA database. The layering definitions are performed entirely within PATRAN using the CSTEM preference. The CSTEM structural input deck is written to the file f35.dat.

The CSTEM heat transfer input deck is written only if heat transfer loads have been applied. Heat transfer deck generation is performed as part of the PATRAN to SIESTA translation process by accessing the PATRAN database only. The CSTEM heat transfer input deck is written to file f34.dat.

A file called WARNINGS is generated which contains informational and warning messages generated during the process of translating the PATRAN database to a SIESTA database and generating the CSTEM input deck from the SIESTA database.

## **GEOMETRY**

The CSTEM preference is based on the HEX8 or HEX20 isoparametric element topology used by the CSTEM finite element code. Only meshes of these element types should be generated from underlying solids (as opposed to curves or surfaces) for use in CSTEM. The solid geometry is created within or imported into PATRAN in whatever manner is familiar to the PATRAN user. Although elements which are invalid within CSTEM can be generated within the PATRAN database, PATRAN will warn the user that such elements are not valid for the active CSTEM preference.

## **MESH GENERATION**

The CSTEM preference does recognize degenerate elements (WEDGE6 or WEDGE15) as valid for use in the CSTEM code. Wedge elements should not be layered in the degenerated direction.

Avoid the generation of elements which are to be layered without underlying geometry. Unless a hybrid axis orientation option (IGAX) is used, the underlying geometry is used to associate the layered material reference axis specified for at least one element to all elements associated with that geometry.

## MATERIAL PROPERTIES

Material models supported by the CSTEM preference are linear elastic, plastic, creep, failure, thermal, and damage. Since the CSTEM finite element code calculates structural and thermal results in the same analysis run, the thermal material properties are specified as a material model of a structural analysis in the CSTEM preference rather than specifying these properties as a separate heat transfer analysis.

Specification of a particular set of material properties will not necessarily activate that type of analysis in CSTEM. Only the linear elastic analysis is performed in CSTEM by default. Inputs for various material modeling capability can be created and present in the input deck without activating that particular type of analysis. The analysis control variables (available on the PATRAN Analysis Form) are used to activate and control the types of analyses performed by CSTEM (see page 21).

Specification of data for material models other than linear elastic will only be written to the CSTEM input deck if the linear elastic data for that material is also specified and can be written. All material model analyses in CSTEM begin with a linear elastic analysis and so linear elastic data for a material is required by the deck generator before any other data for the material will be written.

Many material properties may be input as temperature dependent material tables using the Fields form. Note that PATRAN fields can be specified to calculate extrapolated values as zero, the last table value, or use linear extrapolation. CSTEM always uses linear extrapolation so the material tables are modified if either of the other two PATRAN options (zero and last table value) are specified. These modifications add extra points to the table so that linear extrapolation will result in either zero or the last value. If the addition of extra points will violate the CSTEM limit on number of temperatures or strains, a warning is written and the table is not modified with the result that linear extrapolation will be used. It is recommended that, first of all, material tables be input which bracket the expected ranges and secondly, that linear extrapolation be specified.

### Isotropic

For isotropic materials, the linear elastic properties that can be entered are the elastic modulus, Poisson ratio, thermal expansion coefficient, density, and reference temperature. The elastic modulus, Poisson ratio, and thermal expansion coefficients may be temperature dependent material fields. Common temperature values must be used for these properties for a given material. This will be checked during deck generation.

Inputs to the plastic material model are a stress/strain curve for an isotropic or kinematic hardening plasticity model. Kinematic hardening is the preferred hardening model. The stress/strain curve must be created as a strain or strain and temperature dependent material field prior to its specification as a material property. If specified as a strain and temperature dependent field, the PATRAN field input will require that common strain values be used for all temperatures. This is not a CSTEM requirement, so the CSTEM deck could be edited later to change the strain values at various temperatures if desired.

The isotropic creep model is a 5 coefficient classical creep model, with temperature dependent creep coefficients. Common temperature values must be used for these coefficients for a given material. This will be checked during deck generation. Time or strain hardening is available for use with creep.

Max stress or max strain failure criteria can be used with isotropic materials. The material strengths and strains to failure may be temperature dependent fields. Common temperature values must be

used for these strengths and strains for a given material, which will be checked during deck generation.

The isotropic thermal properties are conductivity and specific heat, which may be temperature dependent fields. Common temperature values must be used for these properties for all materials. This will be checked during deck generation.

Properties for damage mechanics analyses can be entered for isotropic materials, although the available models are typically for use with orthotropic materials. Parameters for the CMCUMAT damage model and/or polynomial damage model may be entered as temperature dependent fields. Common temperature values must be used for all damage parameters of a given material. This will be checked during deck generation.

### **Orthotropic**

For orthotropic materials, the linear elastic properties that can be entered are the elastic moduli, Poisson ratios, thermal expansion coefficients, shear moduli, density and reference temperature. The elastic moduli, shear moduli, Poisson ratios, and thermal expansion coefficients may be temperature dependent material fields. Common temperature values must be used for these properties for a given material, which will be checked during deck generation.

The C.T. Sun orthotropic plasticity model is used with orthotropic materials. This model requires coefficients describing the potential function (used to determine whether the material yields) and the stress/strain curve. The stress/strain curve may be input as a strain dependent field, similar to that for isotropic material, or described using coefficients in a power law function. All potential function coefficients and the stress/strain curve may be temperature dependent fields. Common temperature values must be used for these coefficients for a given material, which will be checked during deck generation.

The C.T. Sun orthotropic creep model is used with orthotropic materials. This creep model utilizes the same potential function as the Sun plasticity model. The potential function coefficients may be entered on either the plasticity form or creep form. The creep curves are described by entering the coefficients of the creep equation. These coefficients may be temperature dependent fields. Common temperature values must be used for these coefficients for a given material, which will be checked during deck generation.

There are 4 failure criteria available for orthotropic materials: Max Stress, Max Strain, Tsai-Wu, and Tsai-Hill. The Max Stress and Max Strain criteria are similar to those for isotropic material, except that strengths or failure strains in all material directions should be entered. The Tsai-Wu failure criteria requires strengths and 3 interaction terms. The Tsai-Hill failure criteria is an in plane criteria, requiring only tensile strengths in the material 1 and 2 directions and in plane shear strength. The failure criteria strengths or failure strains may be temperature dependent fields. Common temperature values must be used for these strengths and strains for a given material, which will be checked during deck generation.

The orthotropic thermal properties are the conductivities in all 3 material directions and specific heat, which may be temperature dependent fields. Common temperature values must be used for these properties for all materials, which will be checked during deck generation.

Parameters for the CMCUMAT damage model and/or polynomial damage model may be entered as temperature dependent fields. Common temperature values must be used for all damage parameters of a given material. This will be checked during deck generation.

## ELEMENT PROPERTIES

The element properties form is used to indicate whether elements are homogeneous or layered. Layered elements are defined as either Layered (manual) or Layered (x-sect) corresponding to the two layering methods available in CSTEM. Layered (manual) indicates that Manual Layer Input will be used and Layered (x-sect) indicates that Cross Section Layup Generation will be used as described in the CSTEM User Manual to define the layering of the designated elements. Somewhat different procedures to specify the layering within PATRAN are used for these two methods.

The integration order of HEX20 elements is also indicated on the element properties form. Standard integration of HEX20 elements utilizes a 3rd order Gaussian quadrature. Reduced integration indicates that a 2nd order Gaussian quadrature should be used. Only 2nd order standard integration can be used with HEX8 elements.

### Homogeneous Elements

There are two element properties for homogeneous elements. Material name is a required element property and material skew is optional. The material specified by material name may be isotropic, orthotropic, or composite material. If a composite material is entered, the equivalent bulk properties of the material will be used. If no material skew coordinate system is specified, the material coordinate system is assumed to be aligned with global (i.e. 1 along X, 2 along Y, 3 along Z).

### Layered Elements

Refer to the CSTEM Users Manual for descriptions of layered elements in CSTEM. These descriptions can be found in Section 2.4.3, Section 4.1.4 and Chapter 5, Section VII.

#### Manual Layered Elements:

Layered (manual) elements have 4 element properties. The layup name is required, similar to the material name for homogeneous elements. The material specified by the layup name may be isotropic, orthotropic, or composite material. If a composite material is specified, the thickness of the material layers must be expressed in decimal fractions of element thickness. Thus, the thickness of the entire layup must add to 1.0, the entire element thickness. If an orthotropic (or isotropic) material is specified as the layup, the material will be oriented along the default material coordinate system generated from the element coordinate system, as determined by the layering definition. In addition to the required layup name, there are three optional element properties: CSTEM variables IGAX and ISMEAR, and a change base axis option.

The hybrid axis option, IGAX, is actually global in scope (the same for all elements) even though as an element property it could be specified differently from element to element. Valid values are 0 to turn off the option, 1, 2, or 3 to orient the material reference axis perpendicular to the projection on the 1-2 material plane of the global X, Y, or Z axis respectively, -1, -2, or -3 to orient the material reference axis parallel to the projection on the 1-2 material plane of the global X, Y, or Z axis respectively. Two digit packed values are also valid to utilize a cylindrical global coordinate system. The two digits indicate the radial axis \*10 + the tangential axis, with the material reference axis oriented perpendicular (+) or parallel (-) to the tangential axis projection based on the sign of IGAX. The first non-zero value in the model found for IGAX will be used. If different values for IGAX are encountered a warning will be printed, but the first non-zero value will be used in the CSTEM input deck. Note that specification of IGAX will eliminate the need to specify the zero degree reference axis using discrete element fields.

The bulk property option, ISMEAR, indicates that the properties of the material layers in the layup for the element are to be combined to generate equivalent orthotropic material properties in the

CSTEM analysis. Valid values for ISMEAR are 0 to not use and 1 to use equivalent bulk properties. The value for ISMEAR may be specified differently from element to element.

The change base axis option relates to the generation of the material coordinate system from the element coordinate system. In CSTEM, the material coordinate system is formed by performing a sequence of cross products on the element coordinate system, using the specified material 3 axis (CSTEM variable LAX) as the base axis by default. The base axis remains unchanged by the cross product process. This default preserves the material 1–3 plane and directly uses the through thickness element coordinate system axis orientation as the material 3 direction. This may be inappropriate for elements skewed through the thickness. In such a case, one of the other two element coordinate system axes, 1 or 2, can be used as the base axis for generation of the material coordinate system. Specifying the base axis as 1 will use the default 0° material reference axis as the base axis. Specifying the base axis as 2 will use the default material in plane transverse (90°) axis as the base axis. Either will preserve the material 1–2 plane and change the orientation of the material 3 axis from the through thickness element coordinate system axis.

It may be advantageous to know the relationship between PATRAN HEX element faces and CSTEM element faces to help with the change base axis specification. The PATRAN face number can be determined from within PATRAN by selecting the element face. Knowing the relationship between the PATRAN element faces and CSTEM element faces, the CSTEM face number can be determined. The CSTEM face number indicates the orientation of the element coordinate system, which is also the initial material coordinate system orientation. In any case it is recommended that the resulting layer orientations be checked using the layer visualization method described on page 26.

The relationship between PATRAN element face numbers and CSTEM element face numbers and the CSTEM coordinate system axes:

PATRAN face number:	1	2	3	4	5	6	
CSTEM face number:	5	6	1	4	2	3	
Element Coord System axis:	+r	-r	+t	-s	-t	+s	
Matl Coord System axis:	+1	-1	+3	-2	-3	+2	(for stacking axis LAX = t)
Matl Coord System axis:	+3	-3	+2	-1	-2	+1	(for stacking axis LAX = r)
Matl Coord System axis:	+2	-2	+1	-3	-1	+3	(for stacking axis LAX = s)

Figure 1 displays the manual layered element property form as it appears in PATRAN.

manual layered solid

Property Name

Value

Value Type

Layup Name	1	Mat Prop Name
[IGAX (hybrid axis)]		Integer
[ISMEAR (bulk property)]		Integer
[Change base axis to:]		Integer

Material Property Sets

pa99uno

0/90/0 layup

0/90 9 ply layup

[90]S

[0]S

OK

Layup Name is required input  
Other three property fields are optional

IGAX should be the same for all elements  
0=off, 1=X, 2=Y, 3=Z  
negative = parallel, positive = perpendicular  
use 2 digit values to indicate radial/tangential

Enter 1 to calculate & use bulk properties

Basis axis for formation of matl coord system from elemental coord system.

Basis axis orientation is unchanged in matl system

Default is through thickness (3) axis.

Enter 1 or 2 to use ref or transverse axis as basis

Layup Name  
thicknesses must add to 1.0

Figure 1. Manual Layered Element Properties Form



### Cross Section Layered Elements:

Layered (x-sect) elements have 4 element properties, all of which are optional. There is no layup specification since this is done by assignment and definition of generation orders as determined from the layering definition. Three of the element properties are identical to those for manual layered element properties: CSTEM variables IGAX and ISMEAR, and the change base axis option. The third element property indicates whether the cross section layup should be applied symmetrically or not. This element property is important for those elements (usually surface elements) to which the spatial FEM field is applied, defining the generation order to be used for the cross section. Valid values for this flag are 0 for a symmetric layup (the default) or 1 for a non-symmetric layup.

Figure 2 displays the Cross Section Layered Element Properties form as it appears in PATRAN.

### **Unspecified Element Properties**

Although not a recommended practice, but possibly occurring due to oversight, elements with element properties left unspecified will be assumed to be homogeneous with integration order 2. The material type of unspecified elements will be output as material number 0, which defaults in CSTEM to material 1. Elements with layered element properties (manual or cross section) will not be layered if the layering definition (Fields) has not been supplied for those elements. For manual layered elements with no layering definition, the element material number will correspond to the material given as the layup name, while for cross section layered elements it will be 0. The integration order will be as specified by the Standard or Reduced Integration menu selection.

Input Properties		
x-sect layered solid	Property Name	Value Type
	[IGAX (hybrid axis)]	Integer
	[ISM/EAR (bulk properties)]	Integer
	[unsymmetric layout (-1)]	Integer
	[Change base axis to:]	Integer

IGAX should be the same for all elements  
0=off, 1=X, 2=Y, 3=Z  
negative = parallel, positive = perpendicular  
use 2 digit values to indicate radial/tangential

Enter 1 to calculate & use bulk properties

Enter 1 to apply defined layout  
through entire cross section.  
Default is to apply symmetrically

Basis axis for formation of matl coord system  
from elemental coord system.  
Basis axis orientation is unchanged in matl system  
Default is through thickness (3) axis.  
Enter 1 or 2 to use ref or transverse axis as basis

Figure 2. Cross Section Layered Element Properties Form

## LAYERING DEFINITION

The CSTEM preference supports both the manual layering and cross section layering methods used by the CSTEM program. Manual layering requires the layup be completely defined and assigned to the element. For manual layering, layer thicknesses are in terms of element thickness fractions and thus add to 1.0. Cross section layering requires that repeating layer subsets and the order that these subsets are applied be defined and applied to model cross sections. (A model cross section is a through thickness sequence of elements.) For cross section layering, layer thicknesses are in actual thickness dimensions (i.e. inches). The alternative method of assignment of a representative cross section to other model cross sections is not available from the CSTEM preference.

Definition of layering using the CSTEM preference involves 4 steps, which are listed below with the Menu locations where the step is performed. These steps do not necessarily have to be performed in the order cited, but substeps do (i.e. substep 1a must be performed before substep 1b).

- 1) a) Specification of individual layer material properties  
(Materials: Create/Isotropic/Manual or Create/Orthotropic/Manual)  
b) Definition of composite material layers comprising generation sets  
(Materials: Create/Composite/Laminate)
- 2) Definition of element properties:  
layered or homogeneous, IGAX, ISMEAR, unsymmetric layup, basis change  
(Element Props: Create/3D/Solid or Modify/3D/Solid)
- 3) Definition of layering fields:  
stacking axis, generation order, and reference axis  
(Fields: Create/Spatial/FEM, Discrete, Scalar, Element)
- 4) Specification of generation sets and definition of generation orders  
(Analysis: Write Input Deck/Entire Model/Full Run)

### Step 1:

The specification of individual layer material properties must be done before a composite layup can be defined. The layer material properties may be isotropic or orthotropic. The specification of a composite layup should be done using the Laminate method. The composite material layup is used differently depending whether manual or cross section layering is used. For manual layering, the sum of the layer thicknesses must add to 1.0 since the thickness for the manual layering method is in terms of the element thickness. For cross section layering, the layer thicknesses are the actual thickness dimensions and the defined material layup will be used as a repeating set of layers in a sequence of such sets. The stacking sequence convention specification on the laminate composite form must also be considered since use of one of the symmetry conventions will double the number of layers in the layup. In the case of cross section layering, this symmetry applies to the repeating set of layers only.

### Step 2:

Elements are defined as layered (manual), layered (x-sect), or homogeneous through assignment of element properties. Element properties can be modified after creation, but changing between homogeneous elements and layered elements may not be possible. Element properties are further described in the Element Properties section of this document.

#### Homogeneous Elements:

Elements defined as homogeneous are not layered and are integrated using a standard Gauss quadrature. The element properties for homogeneous elements consist of a mandatory material name and

an optional material skew coordinate system reference. These elements can be made up of isotropic or orthotropic material, depending on what material is assigned to them. Homogeneous elements which are assigned composite materials will use the equivalent bulk (smeared) orthotropic material properties calculated from the composite layup assigned to the element. Orientation of the material corresponds to the local skew material coordinate system assigned as an element property. If no skew material coordinate system is assigned, the material is aligned with the global coordinate system: 1 along X, 2 along Y, and 3 along Z.

#### Manual Layered Elements:

Layered (manual) element properties consist of a mandatory layup name and three optional properties: the IGAX hybrid axis specification, the ISMEAR equivalent bulk property indicator, and the change base axis option. The Element Properties form for manual layered elements is shown in Figure 1. The layup name is a required property and may be an isotropic or orthotropic material as well as a composite laminate material. If an isotropic or orthotropic material is specified as the layup, the element will have a single layer and thus be homogeneous. The orientation of the material will be defined by the thickness and reference axes of the element specified by the field of Step 3.

The IGAX variable is currently a global variable. It applies to and is the same for all elements in the model and so should be specified the same for all elements. However, as an element property, different IGAX values can physically be specified for different elements. *Only the first encountered non-zero value for IGAX will be used.* Absolute values of 1, 2, or 3 are valid for the IGAX material property, corresponding to the global X, Y, and Z axes respectively. A two digit combination of 1, 2, or 3 is also valid for IGAX, indicating a radial/tangential orientation. Activation of IGAX in CSTEM will rotate the material reference direction (0° direction) about the material thickness axis until it is oriented with respect to the projection of the IGAX global axis on the material 1-2 plane. The material reference direction is oriented with respect to the tangential (ones digit) axis for two digit radial/tangential combinations. Positive values of IGAX indicate that the material reference direction is to be perpendicular to the IGAX global axis, negative IGAX values indicate that the material reference direction is to be parallel to the IGAX global axis.

The ISMEAR option indicates the desire to use equivalent bulk properties, and is activated in the CSTEM input deck. Values of 0 or 1 are valid for the ISMEAR element property. Setting the ISMEAR element property to 1 indicates that equivalent bulk properties should be used, while 0 indicates that explicit layered properties should be used. The default, if no value is entered for ISMEAR, is ISMEAR=0.

The change base axis option indicates a change in the default sequence of cross products performed on the element coordinate system used to generate the material coordinate system. The default sequence preserves the 1-3 plane as defined by the element coordinate system and leaves the specified through thickness axis unchanged. For elements which are skewed in the thickness direction, the change base axis option allows the generation of the material coordinate system to be modified so that it is oriented properly in the thickness direction.

Layered (manual) elements which are assigned a layup (which can be a defined composite material or an isotropic or orthotropic material) will be treated as homogeneous if no other steps which further define the layering are performed. Since no skew material coordinate system can be assigned to layered (manual) elements, the material will be aligned with the global coordinate system: 1 along X, 2 along Y, and 3 along Z.

### Cross Section Layered Elements:

Layered (x-sect) elements have no mandatory element properties. The four optional element properties are the IGAX hybrid axis specification, the ISMEAR equivalent bulk property indicator, the unsymmetric laminate option, and the change base axis option. The IGAX, ISMEAR, and change base axis element properties are similar to those for manual layered elements. The Element Properties form for cross section layered elements is shown in Figure 2. Layups are not specified as a cross section layered element property, but are specified as described in Step 4.

The IGAX variable is currently a global variable. It applies to and is the same for all elements in the model and so should be specified the same for all elements. However, as an element property, different IGAX values can physically be specified for different elements. *Only the first encountered non-zero value for IGAX will be used.* Absolute values of 1, 2, or 3 are valid for the IGAX material property, corresponding to the global X, Y, and Z axes respectively. A two digit combination of 1, 2, or 3 is also valid for IGAX, indicating a radial/tangential orientation. Activation of IGAX in CSTEM will rotate the material reference direction (0° direction) about the material thickness axis until it is oriented with respect to the projection of the IGAX global axis on the material 1-2 plane. The material reference direction is oriented with respect to the tangential (ones digit) axis for two digit radial/tangential combinations. Positive values of IGAX indicate that the material reference direction is to be perpendicular to the IGAX global axis, negative IGAX values indicate that the material reference direction is to be parallel to the IGAX global axis.

The ISMEAR option indicates the desire to use of equivalent bulk properties, and is activated in the CSTEM input deck. Values of 0 or 1 are valid for the ISMEAR element property. Setting the ISMEAR element property to 1 indicates that equivalent bulk properties should be used, while 0 indicates that explicit layered properties should be used. The default, if no value is entered for ISMEAR, is ISMEAR=0.

The unsymmetric layup option is used to indicate whether the model cross section should be layered symmetrically or not. This corresponds to the sign of the CSTEM variable LAX. Values of 0 and 1 are valid for the unsymmetric layup option, with 0 indicating the layup is to be generated as symmetric and 1 indicating the layup is not symmetric. The default, used if no value is indicated for this option, is to indicate a symmetric layup (0).

The change base axis option is used to modify the manner in which the material coordinate system is derived from the element coordinate system. The material coordinate system is derived by performing a sequence of cross products performed on the element coordinate system. Normally this is done so that the orientation of the element coordinate system axis corresponding to the through thickness stacking axis (i.e. material 3 axis) is not changed. For elements skewed in the thickness direction of the mesh, this is not appropriate. The stacking axis should be perpendicular to the plane of the material layers. In this situation, the change base axis option allows the user to specify that an element coordinate system axis other than the one corresponding to the stacking axis be used as the base axis for generating the material coordinate system. The orientation of the user specified base axis would not be changed, allowing the stacking axis to be adjusted so that it is perpendicular to the material layers. The base axis is 3 by default. Setting the base axis to 1 or 2 changes this default. Changing the base axis to 1 indicates that the element coordinate system axis corresponding to the material reference axis is to be used as the base axis. Changing the base axis to 2 indicates that the other in plane element coordinate system axis, the one corresponding to the material 90° axis, is to be used as the base axis. Either of these two axes could be used, although the skewness of the mesh in the plane of the material layers may determine which axis should be specified.

Layered (x-sect) elements for which no other steps are performed to further define the layering will be assigned a material number of zero in the CSTEM input deck. CSTEM translates a material number zero as material 1, so if a material 1 is written to the CSTEM input deck the layered (x-sect) elements will act as homogeneous elements with material 1 properties. Since no skew material coordinate system can be assigned to layered (manual) elements, the material will be aligned with the global coordinate system: 1 along X, 2 along Y, and 3 along Z.

### Step 3:

Layering of elements in CSTEM is generally done with reference to the element coordinate system. This requires specification of the stacking (through thickness) axis and the reference (0 degree) axis of the element. This specification is done utilizing the FEM method of the PATRAN Spatial Field option, applying discrete scalar fields onto element faces. The Fields form is shown in Figure 3 with the appropriate settings for Action/Object/Method and the toggles.

#### Stacking Axis:

The stacking face of an element is identified by applying a field value between 0 and 99 on the element face. The stacking face of an element is a face normal to the stacking axis (i.e. through thickness axis) of the layup contained within the element. The stacking face is in the plane of the first or last layer in the element, so either of two faces can be specified as the stacking face of an element. Multiple specifications for an element will cause any previous specifications for the same element to be overwritten. The same stacking axis will be identified regardless of which of the two stacking faces the field is applied to. Figure 4 shows the field table data form ready to identify selected element faces as stacking faces.

For layered (manual) elements, the stacking sequence progresses from the face onto which the field is applied. This means that the first layer defined in the composite material will lie at the face onto which the discrete scalar field is applied and subsequent layers will be stacked in sequence in the direction towards the opposite face of the element.

For layered (x-sect) elements, only one element in a cross section should have a stacking face specified. A cross section is a sequence of elements along the stacking axis through the thickness of the model. CSTEM input does allow for specification of sub-surface cross sections, but the capability to specify sub-surface cross sections does not exist in the current preference. Therefore, cross sections as identified by the preference are assumed to begin at a free surface and continue through the thickness of the mesh to the opposite free surface. This being the case, it is good practice to apply the field defining the stacking axis to the surface face of an element at the mesh surface. Note that the preference does not check whether or not the field has been applied to a free surface.

The image shows a screenshot of a software form titled "Fields Form". The form contains several input fields and buttons. The "Action" field is set to "Create". The "Object" field is set to "Spatial". The "Method" field is set to "FEM". There are two "Existing Fields" listed: "layer" and "pressure". The "Field Name" field is set to "used layer". The "FEM Field Definition" has two radio buttons: "discrete" (selected) and "continuous". The "Field Type" has two radio buttons: "Scalar" (selected) and "Vector". The "Entity Type" has two radio buttons: "Node" and "Element" (selected). There is an "Input Data" button. At the bottom right, there is an "Apply" button.

Figure 3. Fields Form Settings for Layering Fields

Select an Element  
Elem 1,2,3,4,5,6,7,8,9

Entity	Values
1	
2	
3	
4	
5	
6	
7	
8	
9	

Delete selected row(s)  
Clear selected cells  
Number of rows to insert: 1  
Insert row(s)  
OK

Figure 4. Stacking Axis Assignment

Input Data  
1

Entity	Values
1	Elem 1,2,3,4,5,6,7,8,9
2	Elem 1,2,3,4,5,6,7,8,9
3	Elem 1,2,3,4,5,6,7,8,9
4	Elem 1,2,3,4,5,6,7,8,9
5	Elem 1,2,3,4,5,6,7,8,9
6	Elem 1,2,3,4,5,6,7,8,9
7	Elem 1,2,3,4,5,6,7,8,9
8	Elem 1,2,3,4,5,6,7,8,9
9	Elem 1,2,3,4,5,6,7,8,9

Delete selected row(s)  
Clear selected cells  
Number of rows to insert: 1  
Insert row(s)  
OK

Figure 5. Generation Order Assignment

#### Generation Order:

Layered (x-sect) elements obtain layup information from a layup generation order (stacking sequence) applied to the cross section. The generation orders themselves are defined in Step 4. The assignment of a particular generation order to a cross section is defined by the value of the field applied to an element face in the cross section. Currently, CSTEM allows a maximum of 90 generation orders, so the field value should be from 1 to 90, corresponding to the number of the generation order that is to be used to define the cross section layup. Layered (manual) elements obtain complete element layup information from the composite material definition specified as the layup name for the element. Since layered (manual) elements do not utilize generation orders, any number between 0 and 99 can be used to specify the stacking axis, but a value of 0 is recommended. Figure 5 shows the field table data form ready to assign a generation order to element stacking faces.

#### Reference Axis:

The axis from which the layer orientation is measured is called the reference axis. Specification of a stacking axis implies a default reference axis in CSTEM. Since the default reference axis is not easily determined by the PATRAN user, this axis must also be indicated by assignment of a spatial FEM discrete element field. The reference axis is identified by applying a field value of 100 or greater to the face normal to the reference axis. If the specified reference axis does not correspond to the default, an angular offset of 90 degrees is used with the orientations of the layers within the cross section layup. The reference axis and stacking axis field values can be applied within the same or in separate spatial FEM fields.

The stacking axis and reference axis are based on element coordinate systems, which may vary from element to element. However, a common element connectivity ordering sequence is generally used

within a geometric entity. If all elements within a particular geometric entity have the same reference axis it is undesirable to require that both the stacking axis and reference axis for each element be specified. Thus, it is only required that the stacking axis be specified for every element or cross section, but the reference axis can be inferred by specifying it for as few as one element within a geometric entity. If at least one element in a geometric entity has a specified reference axis, any elements without a specified reference axis that are associated with that geometric entity will use the same reference axis as the specified element. Multiple reference axis specifications can be applied if desired. The first encountered reference axis specification, which will be for the lowest numbered element in the geometric entity with a reference axis specification, will be used for unspecified elements. Explicit reference axis specifications will not be changed by this inferred reference axis capability.

Use of the IGAX hybrid axis option eliminates the need to specify the reference axis since the reference axis will be oriented relative to the IGAX global axis.

#### **Step 4:**

Generation sets and generation orders must be defined for use with layered (x-sect) elements. Generation sets are sequences of material layers and can be thought of as the repeating sets of layers used as building blocks to specify a layup stacking sequence. Generation orders are sequences of generation sets and define the actual layup stacking sequence. Generation sets are the composite material definitions of Step 1b.

Generation sets and generation orders are defined using the forms accessed from the 'Specify Layer Gen Sets' button of the Analysis form. Selection of this button displays two forms, which are shown in Figure 6. The form labelled 'Select Layup Generation Sets' lists all the composite materials defined in the PATRAN database. The form labelled 'Generation Order Definition' allows definition of the generation order number specified in the generation order number databox. This generation order will be used to layer cross sections with layering field values matching the generation order number. Composite materials to be used as generation sets are highlighted one at a time and the Apply button of the 'Select Layup Generation Sets' form is selected to add a generation set to the generation order. The selected generation set is assigned an appropriate number for use in the CSTEM input deck and appears in the generation order listbox. Generation sets are added to the generation order one at a time until the generation order is completed as desired by the user. The Apply button on the 'Generation Order Definition' form is then selected to store the generation order definition. An individual generation set can be deleted from the generation order definition using the Delete button. The entire generation order listbox can be erased using the Reset button. The Apply button must be selected to store any changes made to a previously saved generation order.

The preference will generate two generation sets on the CSTEM input deck for every generation set defined. One is the original generation set with orientations as entered by the user. The second contains orientations with a 90 degree offset, which are used if the specified reference axis does not correspond to the default. The CSTEM preference will similarly create a generation order using the original generation sets and the rotated generation sets. The preference determines the proper generation order to use in the CSTEM input deck from the reference axis information of a cross section specification.

Note that generation orders are not stored in the PATRAN database, but only in memory. Since this information may take some time to generate, the information is written to a version numbered file called GEN.DEFN, which can be used to restore previous generation order definitions. Additional



## LOAD CASES

The CSTEM finite element code has multiple load case capability, but the CSTEM deck generator will translate only a single load case at a time. Multiple load case input decks can be formed by selecting Add Load Case or Add LC (no matl) as the Analysis form method. If this is done in immediate succession to generation of the first load case using the Full Run method, the additional load cases will automatically be appended to the CSTEM input deck file, f35.dat. If done at a later time when the CSTEM input deck file, f35.dat, has been renamed and does not exist, the additional load cases will be written separately to f35.dat and the user will have to manually append this file to the CSTEM input deck. In either case, the CSTEM analysis control variable NLC indicating the number of load cases will have to be changed to the appropriate value.

The difference between the Add Load Case method and the Add LC (no matl) method pertains to whether or not the material properties on the PATRAN database are written to the CSTEM input deck. Material properties can be changed from load case to load case in CSTEM. When material properties are input for a load case, the stiffness matrix will be reformed due to the change in material properties. Reforming the stiffness matrix will take some time, which is unnecessary if the material properties have not really changed. If the material properties on the PATRAN database have not changed, the Add LC (no matl) method should be used to write additional load cases to the CSTEM input deck. If material properties have changed, the Add Load Case method should be used so that the new material properties are written to the CSTEM input deck.

Boundary conditions from multiple PATRAN load cases can be combined into a single CSTEM load case. Input is required to indicate which of the existing PATRAN load cases are to be translated. This information is specified with the Select Load Cases button of the Analysis form. The load cases desired on the CSTEM deck are highlighted in the select databox. A limit exists on the number of load cases that can be combined. Currently this limit is set at 10. This number can not be exceeded since when 10 load cases have been specified for translation, the translation will begin.

In PATRAN, load sets associated with a load case are referenced by ID numbers. A limit exists on the number of load sets that can be translated from a single load case. Currently this limit is set at 100. If the number of load sets in a load case exceeds this limit, a warning is written and the load case is skipped and not translated. PATRAN load cases can be static, time dependent, or frequency dependent. CSTEM can use time dependent heat transfer load cases, so static and time dependent load cases are translated. Static and time dependent heat transfer loads will be written to the CSTEM heat transfer input deck, but only static structural loads are utilized by CSTEM.

Generation Order Definition

Generation Order Number = 1

Generation Sets

Layered Materials

manual\_layup  
quasi-iso  
[0,90]

Repeat from selected Gen Set

Apply Cancel

Delete Reset

Figure 6. Specify Generation Sets Form

details for reading in generation set and generation order definitions are given in the description of the Analysis form (see page 24).

### Layering Recommendations

Since underlying geometric entities can be utilized to implicitly specify a reference axis for elements associated with the geometric entity, it is recommended that elements not be created from manipulation of other elements. Rather, the underlying geometric entities should be manipulated and meshed similarly. However, if the IGAX option will be used this recommendation does not apply.

When applying the spatial FEM fields, consider the posting of individual groups to help avoid the inadvertent application of fields on faces of undesired adjacent elements. There may be times in setting up fields when a mesh seam is bordered by several elements and picking a large number of faces would be easily done using a box or CTRL-pick polygon except that the faces are shared by elements that should not be selected. It may be easier to create a group of the desired elements, posting only that group, then selecting the faces, rather than individually picking faces or editing the picklist to eliminate undesired element faces.

## LOADS AND BCS

PATRAN loads are separated into static, vector, and dynamic loads. Only static and vector structural loads are translated. Dynamic loads are read for heat transfer loads only since CSTEM heat transfer loads can be time dependent. Only loads for 3D elements are translated.

PATRAN allows the specification of a load scale factor, typically located at the top of the specific boundary condition form. The boundary condition values written to the CSTEM input deck are the boundary condition values multiplied by the scale factor.

PATRAN loads can be assigned priorities. These priorities are currently not utilized by the translator. Only the last of multiple similar type loads applied to the same node or element will be applied in the CSTEM load case. These priorities apply within a single load case only and since the capability to combine multiple load cases is being allowed, these priorities lose significance.

Structural loads are displacement, force, pressure, temperature, velocity (RPM), and inertial loads. Heat transfer loads are prescribed temperatures, convection, radiation, surface and nodal heat flux, and internal heat generation. Both types of loads are specified from the Load/BC form of PATRAN. The CSTEM preference restricts the application of these loads to entities that are supported in CSTEM. These entities are nodes and element faces, with the option for uniform element face loads and variable element face loads in some cases.

Boundary conditions applied to nodes are temperatures, displacements, rotational velocities (RPM), linear accelerations, forces, thermal temperatures, and heat fluxes. Thermal temperatures are interpreted as prescribed temperature boundary conditions in CSTEM heat transfer. Initial temperatures are taken by CSTEM as the input structural nodal temperatures. Element temperature specification takes priority over nodal temperature specification in the preference. Rotational velocities are defined as a nodal quantity, so PATRAN will allow different nodes to have different rotational velocities. However, rotational velocities will not be translated to the CSTEM input deck unless applied to all nodes in the model. The same is true for linear accelerations.

Boundary conditions applied to element faces are pressures, convections, radiations, heat fluxes, and heat generation. Pressure loads can be applied as varying over the element face, but are averaged and translated as constant face pressures by the deck generator. Element temperatures have higher priority than a temperature specified as a nodal temperature. Element temperatures can be applied as varying over the element face.

Multiple load variables can be associated with a given load type in PATRAN. Of the recognized static loads, only convection and radiation loads contain more than one load variable. For convection, load variable 1 is the convection coefficient and load variable 2 is the ambient temperature. For radiation, load variable 1 is the radiation (i.e. environment) temperature, load variable 2 is the viewfactor, load variable 3 is the emissivity scale factor, load variable 4 is the gaseous emissivity, and load variable 5 is the gaseous absorptivity. The emissivity scale factor (variable 3) is included as a load so that modifications based on loading can be included in the emissivity, which is entered as a heat transfer material property. The emissivity scale factor multiplies the temperature interpolated emissivity value.

Radiation loads in CSTEM can include modifications for gaseous emissivity and absorptivity. If values are specified on the boundary condition form for these quantities, the modification will be included on the CSTEM input deck, otherwise this modification is not activated. Specifying gaseous emissivity and absorptivity as 1.0 is equivalent to deactivating this modification.

## ANALYSIS

The Analysis form provides access to CSTEM specific functions. The Action, Object, and Method buttons modify what is done from the form. Available options for Action are Write Input Deck, Results, and Write Input Deck (db closed). The Analysis form is shown in Figure 7.

The Analysis form is where the layering definitions are finalized for cross section layered elements, by selecting the Specify Layer Gen Sets button. Details relating to layering definition and the use of this function are described in a separate section of this document, in particular as Step 4 of the layering definition process (see page 16). Other Analysis form functions related to layering are restoration of previous generation set specifications, visualization of layer orientations, and selection of layering fields. Other functions relate to specification of CSTEM analysis and load case control variables and selection of load cases to be translated.

### Action

The Write Input Deck and Write Input Deck (db closed) actions perform exactly the same function, which is to generate a CSTEM input deck. The only difference is that the PATRAN database will be closed during the CSTEM deck generation process if the (db closed) option is selected. The CSTEM deck generation process itself opens the PATRAN database and if all available PATRAN seat licenses are in use this may cause failure of the process. The (db closed) option will have only one database open at a time during CSTEM deck generation. Since it may take extra time to open and close the PATRAN database, use of the (db closed) option is not desirable unless necessary.

Results is a post-processing action. Post-processing of CSTEM results in PATRAN is done by using CSTEM generated neutral results files. Some results files can be translated directly into the PATRAN database, while other results files can be read into PATRAN by selecting File/Import... from the main menubar. The results files are written when the IPAT input variable is activated. A method which creates PATRAN XYDATA files from the CSTEM printed output FILE4 is also available.

### Object

Entire Model and Current Group are available objects for Write Input Deck. Selecting Entire Model will include all appropriate FEM entities in the database into the CSTEM input deck. Selecting Current Group will include only the FEM entities in the currently selected group into the input deck.

Figure 7. Analysis Form

## Method

Two options for Method are available for the Write Input Deck action. The Full Run method writes the entire CSTEM input deck, with boundary conditions in the selected load cases written as CSTEM load case 1. If the Add Load Case method is selected, only the items which apply to CSTEM load case input are written. Model geometry, analysis controls, layering definitions, etc. will not be written when Add Load Case or Add LC (no matl) is selected. If the file `f35.dat` is found, the load case inputs will be appended to this file, but the number of load cases variable, `NLC`, will not be changed.

There are three methods available for the Results action. The Translate method reads CSTEM generated results files directly into the PATRAN database. The Template method will generate template files for reading results files into the PATRAN database using `File/Import...` from the main menubar. The Create XYDATA Files method creates files that can be used to generate XY plots in PATRAN from the CSTEM printed results file, `FILE4`. Results across multiple load cases can be displayed for a particular node or element location. The Results action is described beginning on page 27.

## Write Input Deck

The following sections describe the functions of the buttons pertaining to the Write Input Deck actions on the Analysis form.

### Analysis Controls

Selection of the Analysis Controls button posts a form which contains toggles and databoxes for specification of various global CSTEM analysis and restart options. The CSTEM User Manual contains detailed descriptions of these variables. The analysis options are available as toggles, so they are either on or off. Some of these analysis controls have more than one option in CSTEM. Selection of the toggles will activate the most common and/or recommended options for these variables. Figure 8 shows the Analysis Controls form accessed by selecting the Analysis Controls button.

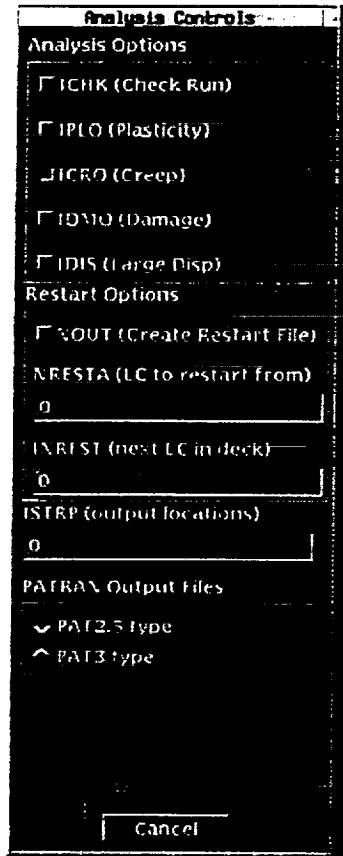
When the creep toggle is activated, an additional form for specifying creep time increment control variables will be posted. Depending on the setting for the number of timesteps, which controls whether dynamic time incrementing is used, another form for specification of the additional variables associated with dynamic time incrementing will appear. Selection of the damage mechanics toggle will cause the time increment control form to be posted, since a non-zero time increment is required for damage to be calculated. The number of timesteps and the dynamic time incrementing form are not applicable for damage mechanics analysis. Figure 10 shows the time increment and dynamic time increment controls forms.

### Load Case Controls

Selection of the Load Case Controls button posts a form which allows specification of variables which can be changed from load case to load case within CSTEM. In addition, a button which can be used to activate a form for input of keyworded input is accessible from this form. The keyword input form simply transfers the string typed in by the user to the keyword input location of the CSTEM input deck without modification. Figure 9 shows the Load Case Controls form accessed by selecting the Load Case Controls button.

### Specify Generation Sets

Generation sets and generation orders for x-sec layered elements are defined by selecting this button. Layering for manual layered elements is defined by specification of a Layup Name on the Element Properties form for manual layered elements (see page 6).



**Analysis Controls**

Analysis Options

☐ ICHK (Check Run)

☐ IPLO (Plasticity)

☐ ICRO (Creep)

☐ IDMO (Damage)

☐ IDIS (Large Disp)

Restart Options

☐ XOUT (Create Restart File)

NRESTA (LC to restart from)

0

INRST (next LC in deck)

0

ISTRP (output locations)

0

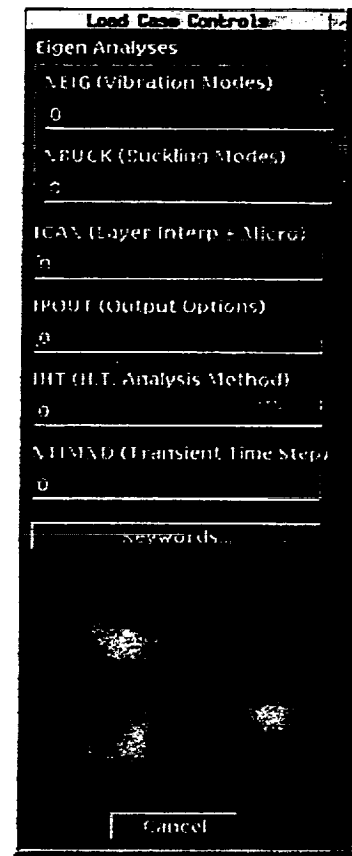
PATRAN Output Files

☒ PAT2.5 type

☐ PAT3 type

Cancel

Figure 8. Analysis Controls Form



**Load Case Controls**

Eigen Analyses

NEIG (Vibration Modes)

0

NEUCK (Buckling Modes)

0

ICAN (Layer Interp + Micro)

0

IPOUT (Output Options)

0

IHT (H.T. Analysis Method)

0

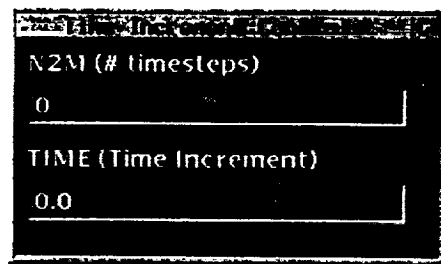
VTIMAD (Transient Time Step)

0

Keywords...

Cancel

Figure 9. Load Case Controls Form



**Time Incrementing Form**

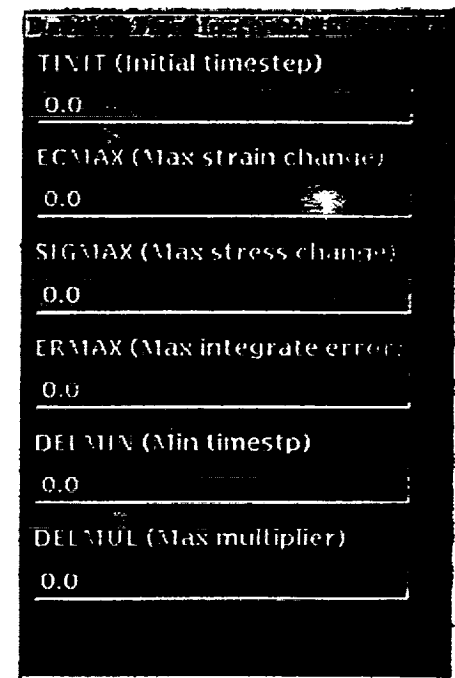
N2M (# timesteps)

0

TIME (Time Increment)

0.0

Figure 10. Time Incrementing Forms



**Time Incrementing Form**

TINIT (Initial timestep)

0.0

ECMAX (Max strain change)

0.0

SIGMAX (Max stress change)

0.0

ERMAX (Max integrate error)

0.0

DEL MIN (Min timestep)

0.0

DEL MUL (Max multiplier)

0.0

Generation sets are defined from composite materials existing in the PATRAN database. Generation orders are defined as a sequence of generation sets. Generation orders define layering of cross sections with field values matching the generation order number. A more detailed description of the use of this form is found on page 16 as Layering Definition, Step 4. The forms displayed by selecting the Specify Generation Sets button are shown in Figure 6.

### Select Load Cases

This button must be selected prior to the Apply button of the Analysis form. If it has not, the Select Load Cases form will be posted and the user will be requested to select one or multiple load cases from the form to be included in the CSTEM input deck. It is not necessary to select a load case, but it is necessary to initialize the form.

All load cases defined in the PATRAN database are displayed in the select databox. Load cases are selected by highlighting the desired load case names. Selection of multiple load cases will combine the boundary conditions of those load cases into a single CSTEM load case. Figure 11 shows the Select Load Cases form accessed by selecting the Select Load Cases button.

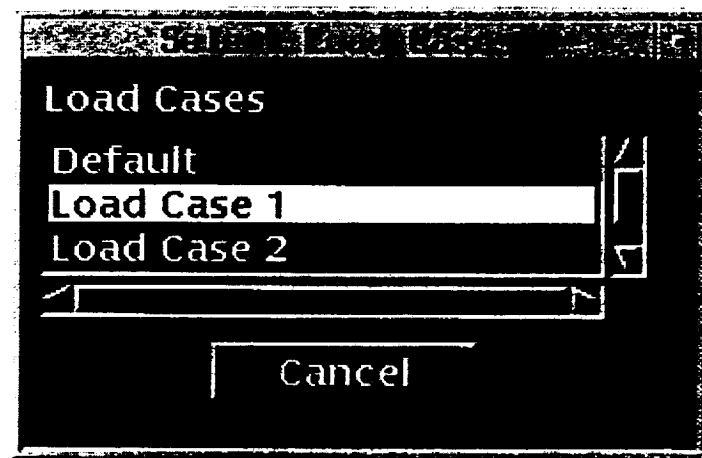


Figure 11. Select Load Cases Form

## Read Layer Gen Sets

Note that generation orders are not stored in the PATRAN database, but only in memory. Since this information may take a significant amount of time to generate, the information is written to a version numbered file called GEN.DEFN. This file contains the generation set and generation order information stored in memory. The generation order information refers to the generation set information, while the generation set information refers to materials stored in the PATRAN database. Selection of the Read Layer Gen Sets button posts a file selection form from which the user can select the desired layering information file to be read. The layering definition file will be read and the layering information stored in memory. It is the users responsibility to make sure that the proper layering information file is used with the PATRAN database which corresponds to the referenced generation set material numbers. Figure 12 shows the file selection form accessed by selecting the Read Layer Gen Sets button.

Read Layer Gen Sets

Filter  
/users/hartle/cstem/HSCT/p3link/G

Directories	File List
/users/hartle/cstem/HSCT/p3link/.	GEN.DEFN.01
/users/hartle/cstem/HSCT/p3link/..	GEN.DEFN.02
/users/hartle/cstem/HSCT/p3link/nasa	GEN.DEFN.03
/users/hartle/cstem/HSCT/p3link/pack_it_up	GEN.DEFN.04

Gen Definition File  
/users/hartle/cstem/HSCT/p3link/

APPLY Filter Cancel

Figure 12. Read Layer Gen Sets file selection form



## Select Layering Fields

Discrete spatial entity fields applied to element faces are used to specify the layering of the model for CSTEM. Generally, all spatial entity fields are scanned to determine if they contain layering definition information. This process involves retrieving the value of the entity field for every face of every element, which can take some time for a very large model. The Select Layering Fields function can be used to indicate only those spatial entity fields which are to be considered for writing to the current input deck. If other spatial entity fields unrelated to layering definition exist in the database, they can be skipped. If several layering definition fields have been created as various alternative layering definitions for the model, the specific field or fields to use can be selected while the alternative layering fields are skipped.

Clicking the Select Layering Fields button on the Analysis form will post the form shown in Figure 13. Fields to be used to define the layering of the model become highlighted when selected. More than one field can be selected.

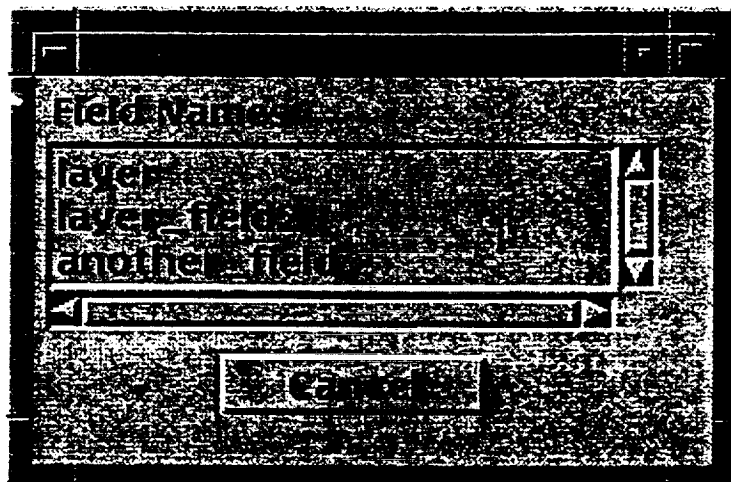


Figure 13. Select Layering Fields form

## Layer Orientation Visualization

Orientation of the fibers in layered elements can be visualized using the Orientation Visualization button of the Analysis form. Layer orientation visualization requires a layer orientation visualization file obtained from a CSTEM check run. The layer for which orientations are desired is entered in the databox, the layer orientation visualization file (CSTEM output file 18) is input, and the Apply button is selected. Fiber orientations are indicated by an arrow placed on the layer midsurface in each element in which the layer occurs. There is a toggle to indicate the choice to exhibit a cross section layer or an element layer. The cross section layer option, which is available for cross section layered elements only, depicts a single layer in each stack of elements. The extent of a particular layer throughout portions of the model having the same generation order can be seen as well as the fiber orientation. In the element layer option, the layer number refers to that layer within every element. Every layered element has a layer 1 even if there are multiple elements through a cross section. The orientation for all layers of all elements can be depicted at once by setting Layer Number = 0 and selecting Apply with the appropriate file entered in the databox. In this case the Element / Cross Section toggle makes no difference. Figure 14 shows the file selection form accessed by selecting the Layer Orientation Visualization button.

Layer Orientation Visualization

Layer Number = 1

Layer Type  
✓ Element    ^ Cross Section

Filter  
/users/hartle/cstem/HSCT/cases/openhole/f

Directories

/users/hartle/cstem/HSCT/cases/openhole/.	/
/users/hartle/cstem/HSCT/cases/openhole/..	/

File List

file+S.d10	/
file+S.disp.i	/
ftn18	/
ftn93	/

Layer Orientation File  
/users/hartle/cstem/HSCT/cases/openhole/ftn18

APPLY    Filter    Cancel

Figure 14. Layer Orientation Visualization Form

## **Results**

There are three methods available for the Results action: Translate, Template, and Create XYDATA Files. Translate and Template are methods to load results for the entire model into the PATRAN database. Create XYDATA Files is a method to generate XY plots of results at a particular location in the model across multiple load cases.

The Translate and Template methods load results for the entire model into the PATRAN database using specific PATRAN results files generated for each solution load case. The nodal displacement and temperature results files, PATRANDISP and PATRANTEMP, can be utilized interchangeably in either the Translate or Template method. The general nodal results file, PATNDRES, is used with the Template method only. The Translate and Template methods utilize different formats for the element based results files, PATELSIG, PATELEPS, and PATDAMGP. The Translate method requires results at every element integration point for either homogeneous or layered elements. The Template method requires results at the centroid of each layer in a layered element. The results files are generated by CSTEM as specified by the IPAT analysis control variable. A positive value of IPAT generates element results files for use with the Template method. A negative value of IPAT generates element results files for use with the Translate method. The other PATRAN results files are generated identically for non-zero IPAT.

### **Translate**

Nodal displacements and temperatures can be translated directly into the PATRAN database from the results files PATRANDISP and PATRANTEMP, respectively. Stress, total strain, and parameter values from the damage mechanics model can also be translated directly into the PATRAN database when element results files PATELSIG, PATELEPS, and PATDAMGP are generated by a negative value of IPAT. When a negative IPAT is specified in CSTEM, results at element integration points are written to these element results files. Results for layered elements are stored in layered form on the PATRAN database. For layered elements the integration points in each layer are located on the plane of the element centroid, so the layer through thickness locations are not available. Integration point results are currently not contour or fringe plottable, but can be used to generate tensor plots and XY plots.

Selecting the Apply button for the Translate method posts the Results Type and Results File Selection forms shown in Figure 15. The results file is selected from the Results File Selection form and the results type contained on the file is indicated by selecting the appropriate toggle switch on the Results Type Selection form. Although several result type toggles are visible for future utilization, only those currently available are selectable.

### **Template**

Appropriate results files can be read into PATRAN by selecting File/Import... from the main menu-bar. A template file describing the format of the results file must be made available to PATRAN in order to read the results file. Selecting the Apply button for the Template method will create PATRAN template files for reading neutral results files written by CSTEM containing displacements (template\_disp), nodal stress/strain results (template\_ndres), element stress (template\_esig10), element strain (template\_eeps10), and element damage parameter results (template\_dam10) for elements with up to 10 layers. The displacement and nodal results template files are applicable to any model, although nodal stress/strain results are not written for layered elements. The element results file can be used with layered or homogeneous elements. Homoge-

**Results Type:**

☐ Model Name   
 ☐ Int.Pt. Stress   
 ☐ Int.Pt. Total Strain   
 ☐ Displacement   
 ☐ Temperature   
 ☐ Damage Parameters

**Filter**  
 /users/hartle/cstem/HSCT/p3link/P

**Directories**

/users/hartle/cstem/HSCT/p3link/.	File List
/users/hartle/cstem/HSCT/p3link/..	PATDAMGP
/users/hartle/cstem/HSCT/p3link/nasa	PATELEPS
/users/hartle/cstem/HSCT/p3link/pack_it_up	PATELSIG
	PATNODELS

**P3 Results File**

Figure 15. Translate Results Selection Forms

**Output FILE4 Selection**

**Filter**  
 /users/hartle/cstem/HSCT/p3link/P

**Directories**

/users/hartle/cstem/HSCT/p3link/.	File List
/users/hartle/cstem/HSCT/p3link/..	
/users/hartle/cstem/HSCT/p3link/nasa	
/users/hartle/cstem/HSCT/p3link/pack_it_up	

**Output FILE4**

**Create XYDATA FORM**

Node	Element

**Element Internal Location**  
 1

**X and Y Results Quantities**

Plot #	X	Y
Plot #1: X1		Y1
Plot #2: X2		Y2
Plot #3: X3		Y3
Plot #4: X4		Y4

**Valid Results Quantities:**

Forces/Disps:	Fx, Fy, Fz, Dx, Dy, Dz
Stresses:	Sig1, Sig2, Sig3, S45, S67, S90
Total Strains:	Eps1, Eps2, Eps3, E45, Eps7, Eps
Damage Strains:	Epd1, Epd2, Epd3, Epd4, Epd5, Epd

Figure 16. Create XYDATA File Forms

neous elements will have results for one layer only. It will be necessary to modify the element results template file if the maximum number of layers in any element is greater than 10.

The nodal results files which can be postprocessed using the template files are PATRANDISP and PATNDRES. PATRANDISP contains nodal displacements, while PATNDRES contains nodal stresses, total strain, failure criteria, and temperatures. Nodal results are fringe plottable.

Element results files PATELSIG, PATELEPS, and PATDAMGP can be postprocessed using template files if generated by a positive value of IPAT. When a positive IPAT is specified in CSTEM, results at element layer centroids are written to these element results files. Element results postprocessed in this way are fringe plottable by averaging the constant element values at shared nodes.

### **Create XYDATA Files**

This method collects results for a particular model location from the CSTEM printed results output file, FILE4, across multiple load cases. Up to four PATRAN XYDATA plot files for certain results quantities can be generated at a time. The XYDATA files are written to the files PATXY . 1, PATXY . 2, PATXY . 3, and PATXY . 4. Once generated, these files can be externally renamed and the process repeated to generate other files for different results quantities or model locations.

Currently, results quantities available for access with this method are the 3 displacement components at a single node, the 3 components of the summation of reaction loads, and the 6 components of stress, total strain, and damage strain at a particular element location. The specific nodal location is simply selected directly from the model. The elemental location is specified by selecting an element from the model, then entering the location within the element where the results are desired. The location within the element corresponds to a line on the output FILE4 for the selected element and as such is either an integration point, element local node, or element face as appropriate for the particular FILE4 being used. So for a homogeneous element with results printed at order 2 integration points any single integer from 1 through 8 would be a valid location.

Besides the XYDATA files, this method also automatically creates four other files which contain all the available results quantities in column form. These files are POST4 . 1, which contains nodal displacements for the specified node and reaction force sums, POST4 . 2, which contains stress results for the specified element location, POST4 . 3, which contains total strain results for the specified element location, and POST4 . 4, which contains damage strain results for the specified element location. These files are not directly usable in PATRAN, but may be utilized as desired by the user.

Selecting Apply for the Create XYDATA Files method on the Analysis form will display a file selection form where the CSTEM printed output FILE4 is selected. Selecting Apply on the output FILE4 selection form causes another form to be displayed. This second form is where the location and results quantities to be used to create the XYDATA files are specified. The two forms used with the Create XYDATA Files method are shown in Figure 16.

## P3 DATABASE TO SIESTA RDB TRANSLATOR

The P3 to SIESTA translator (pdb2rdb.x) can be activated as a standalone program or from within PATRAN. When run as a standalone program, the required user supplied inputs are the database name and a flag indicating the desire to use a particular PATRAN load case (1) or not (0), and values for 24 CSTEM analysis and load case control variables. When run from within PATRAN, these inputs are supplied to the translator program through a script, wrdeck, written and executed by the CSTEM preference.

Multiple load cases can be combined into a single SIESTA (CSTEM) load case. Overwrite for the SIESTA database is turned on during the entire process. BTABs are set to store direct heat transfer values as opposed to table numbers. Blank common IA is used to store node and element IDs, etc. since the process of retrieving these from the PATRAN database generally requires that they all be retrieved at once, returning in a single vector. Currently the length of blank common IA is set to 600000, which implies a general limit of 600000 nodes and 100000 elements. The 100000 element limit is due to the use of blank common to store element face boundary conditions.

The basic functions are:

- 1) open the PATRAN database,
- 2) initialize the SIESTA database,
- 3) check max node and element names in the PATRAN model and increase NLIM if necessary,
- 4) translate coordinate frames,
- 5) translate all NODEs in the PATRAN model, initializing temperatures to 70°F and setting ICS to the PATRAN analysis coordinate system for the node.
- 6) translate all BRI8 or VANS in the PATRAN model, saving material codes,
- 7) translate constraint equations,
- 8) translate material properties (elastic and heat transfer only),
- 9) translate boundary conditions,
- 10) translate CSTEM analysis control variables to SIESTA using ANLS keywords,
- 11) translate the heat transfer material properties and boundary conditions directly to a CSTEM heat transfer input deck,
- 12) 'wrap' the SIESTA rdb and generate 3D surfaces,
- 13) call the CSTEM input deck generator to convert from SIESTA database to CSTEM input deck
- 14) close the PATRAN database.

## FRAMES

PATRAN cartesian coordinate frames are stored in SIESTA as direction cosine transforms (ITYP=4). PATRAN cylindrical and spherical coordinate frames are stored with an associated transformation matrix. Thus, cylindrical and spherical coordinate frames are stored in SIESTA with reference to a direction cosine transformed cartesian coordinate system. Any offsets are translated. The PATRAN coordinate frame IDs are used as the SIESTA CSYD NAME. The associated transform for cylindrical and spherical coordinate systems are stored in SIESTA with NAMES numbered sequentially from the largest PATRAN coordinate frame ID. A limit (blank common size) exists on the number of coordinate frames that can be translated. Since coordinate frames in PATRAN are accessed one at a time, this limit will not be exceeded.

PATRAN unrotated cylindrical coordinate frames have radial along local X and tangential along local Y. SIESTA unrotated cylindrical coordinate frames have radial along local Y and tangential

along local X. Therefore, SIESTA cylindrical coordinate frames correspond to PATRAN coordinate frames such that  $\text{SIESTA } Y = \text{PATRAN } X$ ,  $\text{SIESTA } X = \text{PATRAN } Y$ ,  $\text{SIESTA } -Z = \text{PATRAN } Z$ .

The translation of transformations from PATRAN stores a reference transformation such that global cartesian is transformed to SIESTA cylindrical, but oriented as PATRAN cylindrical. Stated another way, the translation maintains the radial, tangential orientation which is indicated in PATRAN by the local X and Y axes respectively, but stores this orientation using the SIESTA convention with radial along local Y, tangential along local X, and axial along the negative local Z. Note that a corresponding transformation of vector type boundary conditions is performed prior to storage in the SIESTA database.

## NODES

If the number of nodes exceeds the limit (blank common size) the program will STOP. Node names, coordinates, coordinate system and temperature are stored. Temperatures are initialized as 70°F. Actual nodal temperature values (if they exist) are set during BC translation. Two coordinate systems are returned for PATRAN nodes: a reference coordinate system and an analysis coordinate system. The analysis coordinate system is saved as a SIESTA boundary condition coordinate system (BCSY). Note that if any nodal boundary condition is applied to the node, the coordinate system of the boundary condition takes precedence and will overwrite the stored analysis coordinate system. The reference coordinate system is not translated.

## ELEMS

If the number of elements exceeds the limit (blank common size) the program will STOP. Currently, only hexahedral elements are accepted by the translator. If an element other than a hexahedral element is encountered, the program will STOP. The element name, connectivity (converted to SIESTA/CSTEM convention), and material code are stored as BRI8 if the element has 8 nodes or VANS if the element does not have 8 nodes. Unique material codes are stored in an array along with the linearity code, directionality code, and laminate code corresponding to the material. This array is used later to determine the material codes for which it is necessary to translate material properties.

Materials are stored using the same material number as in PATRAN when possible. Note that any unique combination of material, linearity, directionality, and laminate codes generates a new material, so that if a particular PATRAN material code is used with multiple combinations of linearity, directionality, or lamination it must be stored with a material number other than the PATRAN material code. The number of materials is subject to a limit as set by the parameter MXM, which is currently set to 100. If this limit is exceeded, a warning is written and storage of the material is skipped, but translation of the PATRAN database continues.

The integration order of the element is determined from the PATRAN element formulation option. Standard formulation corresponds to integration order 2 for 8 node bricks and 3 for 20 node bricks. Reduced integration applies to 20 node bricks only as an integration order of 2. In SIESTA, this is stored as NINT for the 20 node VANS elements.

There is currently only 1 element property defined for the CSTEM preference which is stored in the SIESTA database, that being the element skew coordinate system. The element skew coordinate system refers to a PATRAN local coordinate system to be used to orient an orthotropic material assigned to the element. This is stored as general element data 12 (GE12) for the element in the SIESTA database. Other element properties (IGAX, ISMEAR, unsymmetric layup) are associated with layered elements and are not stored in the SIESTA database.

## MPCS

PATRAN explicit MPCs are equivalent to SIESTA/CSTEM constraint equations. PATRAN numbers constraint equations independently from elements. The translated constraint equations are numbered in SIESTA in sequential order beginning after the largest element name encountered. If the number of MPCs exceeds the limit (blank common size) the program will STOP. The actual number of SIESTA MPCs could exceed this limit because PATRAN can have sub-MPCs associated with each MPC. As I understand it, PATRAN allows multiple dependent nodes to be specified in explicit MPCs, then internally generates sub-MPCs of the usual form (one dependent DOF per equation). PATRAN defines 36 different types of MPCs. Some of these MPCs include rigid links, sliding surfaces, pinned joints, etc. Only explicit MPCs are translated. If something other than an explicit MPC type is encountered, a warning will be written and that MPC will be skipped and not translated. If an MPC is specified in a coordinate system other than the global coordinate system, a warning will be written and that MPC will be skipped and not translated. Currently a limit of MXDEP=20 independent terms per constraint equations exists, which is the current limit in CSTEM. If this limit is exceeded, a warning is written and the sub-MPC is skipped and not translated.

## MATL

The unique material codes as determined from the element translation are passed into this routine, along with the material linearity code, directionality code, and lamination code. A limit exists on the number of material combinations that can be translated. This limit sizes the array storing the unique material codes. Currently this limit is set at MXM=100. If more than 100 combinations of materials, directionalities, linearities, and laminate code are used in the model, only the first 100 encountered during element translation will be translated. Note that material ids are checked until all existing materials on the PATRAN database are translated to the SIESTA database whether or not they have been detected during element translation. (They may be used by layering!)

PATRAN materials are arranged in material categories with a linearity code and directionality code. Currently the category (isotropic, 2D or 3D orthotropic or anisotropic) is ignored and this information is obtained from the directionality code (isotropic, orthotropic, anisotropic, or 2D orthotropic or anisotropic). Constitutive models are associated with a material code. Currently active constitutive models (those for which the CSTEM preference provides input capability) are elastic, plastic, creep, failure, thermal, and damage. The model parameters are stored in the SIESTA database using storage locations explicitly for such parameters, as ANLS (analysis code specific) data, or as generic table data. Thermal properties can only be stored as isotropic data. This is partially the reason why the CSTEM heat transfer deck is written directly from the PATRAN database rather than from the SIESTA database. (Thermal boundary conditions are the other reason.)

Isotropic plasticity data is stored on the SIESTA database as PLSD data. No modification or alternate use of the reserved SIESTA database locations was necessary.

Potential function parameters for C.T. Sun orthotropic plasticity and creep are stored as ANLS data. (The ANLS keywords used are described in Table 1.) The C.T. Sun plasticity stress-strain curves are stored as PLSD data. Power law stress-strain curve coefficients A, n, and C are stored in the stress-strain pair data locations 16, 17, and 18.

Isotropic creep model parameters are stored in the SIESTA database as CRPD data. No modification or alternate use of the reserved SIESTA database locations was necessary.



keyword	description
CTPF	C.T. Sun potential function indicator
DAMG	Damage Mechanics model indicator (unused)
DONE	End of input indicator
FORM	Damage Mechanics formulation indicator (unused)
TEMP	Temperature
C12	C.T. Sun potential function constant $C_{12}$
C23	C.T. Sun potential function constant $C_{23}$
C13	C.T. Sun potential function constant $C_{13}$
A44	C.T. Sun potential function constant $a_{44}$
A55	C.T. Sun potential function constant $a_{55}$
A66	C.T. Sun potential function constant $a_{66}$
A12	C.T. Sun potential function constant $a_{12}$
A23	C.T. Sun potential function constant $a_{23}$
A13	C.T. Sun potential function constant $a_{13}$

TABLE 1. ANLS KEYWORDS FOR CSTEM MATERIAL MODELS

C.T. Sun orthotropic creep model parameters are stored as CRPD data. The SIESTA formulation code is set to 5 to indicate C.T. Sun creep. The 7 creep coefficients are stored in the order B, C, D, m, n, r, s.

Failure strengths (or failure strains) are stored in the SIESTA database as STMD data. The unspecified primary data location 8 is used to indicate the type of failure criteria used. The code is identical to that used in CSTEM: 1=Tsai-Wu, 2=max stress, 3=max strain, 4=Tsai-Hill. The strengths (or failure strains) are stored in table locations 17 through 25 in the order FT1, FC1, FT2, FC2, FT3, FC3, S12, S23, and S31 where T denotes tensile and C denotes compressive strengths. Data locations 26, 27, and 28 are used for the Tsai-Wu interaction terms F12, F23, and F31.

Damage mechanics parameters are stored as generic table data (TABL). Vector DTDAMG is used to store the table. Table 2. lists the vector locations of the damage parameters as stored in the table vector DTDAMG.

A number (about 4 pages worth) of generic material words are associated with a material code in PATRAN. Only recognized material words are translated. Currently recognized material words are for elastic moduli (E11, E22, E33), Poisson ratios (PR12, PR23, PR13 or PR31), shear moduli (G12, G23, G31), density, thermal expansion coefficients ( $\alpha_{11}$ ,  $\alpha_{22}$ ,  $\alpha_{33}$ ), thermal conductivities (k11, k22, k33), specific heat (Cp), emissivity ( $\epsilon$ ), and reference temperature. Poisson ratios are assumed to be row normalized (CSTEM convention). Number of plies is also a recognized material word. If other than a recognized material word is encountered, a warning is written and that material word will not be translated.

Material words contain either a constant material property value or a field number referencing a temperature or time varying material field. Both constant valued and field specified properties are translated, although fields must be in tabular form to be translated. Material property fields in PATRAN can also be in PCL equation form and it would be possible to generate a material property table from the PCL equation, but this is not currently done. All recognized generic material words are tempera-

<u>CMCUMAT</u>		<u>Polynomial</u>	
1	Material Number		Material Number
2	Table Type (0)		Table Type 0)
13	Damage Model (2 or 20)		Damage Model (3)
14	Fiber Reference Length, L0		
15	Fiber Radius, R		
16	Composite Angle, $\gamma$ (unused)		
17	Matrix Volume Fraction, fm		
18	Fiber Volume Fraction, ff		
25	Temperature		Temperature
26	Matrix Modulus, Em		Order 1 Cycle Coeff, AA
27	Matrix Poisson Ratio, PRm		Order 2 Cycle Coeff, BB
28	Matrix Weibull exponent, mm		Order 3 Cycle Coeff, CC
29	Matrix Energy Release Rate, Ym		Order 4 Cycle Coeff, DD
30	Matrix Saturation Value, Dsat		Stress Ratio Constant, SA
31	Fiber Modulus, Ef		Order 1 Stress Ratio Coeff, SB
32	Fiber Poisson Ratio, PRf		Order 2 Stress Ratio Coeff, SC
33	Fiber Weibull Exponent, mf		Normalization Stress, SU
34	Fiber Interface Shear Resistance, th0		
35	Fiber Reference Stress, S0		
36	Normal Exponent, mS		
37	Normal Normalization Stress, SNO		
38	Normal Threshold Stress, Sth		
39	Normal Saturation Value, Samp		
40	Shear Exponent, mT		
41	Shear Normalization Stress, TNO		
42	Shear Threshold Stress, Tth		
43	Shear Saturation Value, Tamp		

TABLE 2. DAMAGE PARAMETER LOCATIONS IN SIESTA TABLES

ture varying properties only. A limit exists on the number of temperatures that can be translated in a table. The current limit is 10 temperatures. If a table contains more than 10 temperatures, a warning is written and only the first 10 temperatures of the table will be translated. Stress-strain curve pairs are entered as a function of temperature and strain. The strain is treated as an independent variable, which requires that identical strain values be used at all temperatures. This is a restriction imposed by PATRAN field forms and is not necessary for either the SIESTA database or CSTEM input. A limit of 10 stress-strain pairs exists. If more points than this limit is entered, a warning will be written and only the first 10 points will be translated.

Note that PATRAN fields can be specified to calculate extrapolated values as zero, the last table value, or use linear extrapolation. CSTEM always uses linear extrapolation so the material tables are modified if either of the two PATRAN options (zero and last table value) are specified. These modifications add extra points to the table so that linear extrapolation will result in either zero or the last value. If the addition of extra points will violate the limit on number of temperatures or strains, a warning is written and the table is not modified with the result that linear extrapolation will be used. It is recommended that, first of all, material tables be input which bracket the expected ranges and secondly, that linear extrapolation be specified.

There are 25 generic load types in PATRAN. Only recognized structural and heat transfer load types are translated. Currently recognized generic structural loads are displacement, force, pressure, temperature, velocity (RPM), and inertial loads. There are no CSTEM specific structural loads defined by the CSTEM preference. Thermal temperatures are the only recognized generic heat transfer load. CSTEM specific heat transfer loads that are recognized are convection, radiation, surface and nodal heat flux, and internal heat generation. If unrecognized load types are encountered, a warning is issued and the load is skipped and not translated.

Multiple variables can be associated with a given load type. Vector loads generally contain the translation (force) vector as the first variable and rotation (moment) vector as the second variable. Of the recognized static loads, only convection and radiation loads contain more than one load variable. For convection, load variable 1 is the convection coefficient and load variable 2 is the ambient temperature. For radiation, load variable 1 is the radiation (i.e. environment) temperature, load variable 2 is the viewfactor, load variable 3 is the emissivity, load variable 4 is the gaseous emissivity, and load variable 5 is the gaseous absorptivity. The emissivity (variable 3) is included as a load rather than material property so that modifications based on loading can be included in the emissivity.

The CSTEM preference has been created to restrict the application of loads to entities that are supported in CSTEM. These entities are nodes and element faces, with the option for uniform element face loads and variable element face loads in some cases. Although not possible with the CSTEM preference, load application on element edges is checked in the boundary condition translation process. If element edge loads are encountered, a warning is written and the load is not translated. Eventually, it may be desirable to convert element edge loads to nodal loads and translate them that way.

Boundary conditions applied to nodes are temperatures, displacements, rotational velocities (RPM), and forces. Thermal temperatures are interpreted as prescribed temperature boundary conditions in CSTEM heat transfer. Initial temperatures are taken by CSTEM as the input nodal temperatures. Element temperature specification takes priority over nodal temperature specification, which is done in the following way. Prior to storage of a nodal temperature, the first working space in the NODE storage vector (vector location 7) is checked and if found to be 1 the previous temperature for the node will not be overwritten. Different nodes can have different rotational velocities in PATRAN. The first encountered rotational velocity is stored locally and checked against all other rotational velocities found. If a difference is detected a warning is written, but the new rotational velocities will be translated. This means that the last encountered rotational velocity is what will be translated.

Boundary conditions which are constant and applied to element faces are stored using BTABs. The BTAB NAME used is the same as the PATRAN load ID. Such loads are pressures, convections, heat fluxes, and heat generation. Pressure loads can also be applied as varying over the element face, in which case they must be stored directly (BRI8 or VANS) and not as BTABs. Varying element pressures are averaged and translated as constant face pressures (BRI8 locations 45 – 50, VANS locations 41 – 46). Element temperatures are translated as nodal temperatures with the first working space in the NODE storage set to 1 to indicate the temperature is to have higher priority than a temperature specified as a nodal temperature. Element temperatures can be applied as varying over the element face. Varying elemental convection, radiation, heat fluxes, and heat sources are currently unable to be applied from the CSTEM preference, but coding exists which checks for varying loads of these types and writes a warning that they can not be translated.

Nodal forces and displacements can be applied relative to a local PATRAN coordinate system. The PATRAN coordinate systems are translated to SIESTA as described in the section about FRAMES.

CSTEM requires that the different elastic properties be specified at the same temperature for a given material. The specification temperatures may be different for different materials. As each property is retrieved from the PATRAN database, the specification temperatures are checked against the previously stored specification temperatures. If a difference is detected, a flag is set indicating this fact. After all properties are retrieved, a warning is written and the properties for the material will not be translated. Specification of a constant material property together with other field specified temperature varying material properties is acceptable and the constant value will be translated at the same temperatures given for the varying properties.

The heat transfer properties are even more restrictive in CSTEM in that different heat transfer properties for all materials must be specified at the same temperatures, although they may be different from the structural elastic property temperatures. However, SIESTA allows specification temperature differences between different heat transfer materials. Therefore, the heat transfer properties are checked in the same way as the elastic properties and any differences between materials would have to be handled by the CSTEM deck generator in SIESTA. (However, as previously mentioned, the thermal data is translated to the CSTEM input deck directly from the PATRAN database.)

The elastic material properties are stored in SIESTA as either temperature varying isotropic, temperature varying orthotropic, constant isotropic, or constant orthotropic material tables (DMAT). Heat transfer properties are stored as temperature varying isotropic material tables (HTMD). If orthotropic conductivities are entered, k11 is used.

## BCS

Boundary conditions from multiple PATRAN load cases can be combined into a single stored SIESTA load case. Input is required to indicate which of the existing PATRAN load cases are to be translated. The load case name is referenced and a prompt is issued to enter a 1 (one) if the loads contained in the referenced load case are to be translated or a 0 (zero) if they are not to be translated. Anything other than a 1 (one) is interpreted as a 0 (zero). It would be possible to translate only the active load case, which can be set in PATRAN prior to exiting the program, but this is not currently done. A limit exists on the number of load cases that can be combined. Currently this limit is set at 10. This number can not be exceeded since when 10 load cases have been specified for translation, the translation will begin.

In PATRAN, loads associated with a load case are referenced by ID numbers. A limit exists on the number of load sets that can be translated from a single load case. Currently this limit is set at 100. If the number of load sets in a load case exceeds this limit, a warning is written and the load case is skipped and not translated. PATRAN load cases can be static, time dependent, or frequency dependent. CSTEM can use time dependent heat transfer load cases, so static and time dependent load cases are translated. Static and time dependent heat transfer loads will be written to the CSTEM input deck, but only static structural loads are translated to the SIESTA database.

PATRAN loads are also assigned priorities. These priorities are currently not utilized by the translator. Since SIESTA overwrite is turned on, only the last of multiple similar type loads applied to the same node or element will be applied in the SIESTA load case. These priorities apply within a single load case only and since the capability to combine multiple load cases is being allowed, these priorities lose significance.

PATRAN loads are separated into static, vector, and dynamic loads. Only static and vector structural loads are translated. Dynamic loads are read for heat transfer loads only since CSTEM heat transfer loads can be time dependent. Only loads for 3D elements are translated.

For cylindrical or spherical coordinate systems, reordering of the boundary condition components is necessary along with the coordinate system translation. This is due to the fact that PATRAN defines the cylindrical coordinate system in such a way that the 0 degree radial lies along the global X axis with positive angles measured counterclockwise while SIESTA defines the 0 degree radial aligned with the global Y axis with positive angles measured clockwise. The following component reordering is applied for boundary conditions referred to PATRAN local cylindrical or spherical coordinate systems: SIESTA X = PATRAN Y, SIESTA Y = PATRAN X, SIESTA Z = PATRAN -Z.

## **BSURF**

BTAB loads require the load to be stored as a BTAB and the element face to record the BSURF. Combinations of multiple load types on a given element face are permissible, but require a unique BSURFNAME from element faces containing any of the single BTAB loads or a different combination of BTAB loads. This is done by storing BTAB NAME codes in the blank common vector, with the position in the vector indicating the element, face combination. The storage location in the blank common vector is calculated as  $(FACE-1)*MAXEL+ELEM$ , where FACE is the face number (1 to 6), ELEM is the element number, and MAXEL is the maximum allowable element name in the model (currently 100000). This is the source of the previously mentioned current effective limit of 100000 elements for the 600000 blank common vector length.

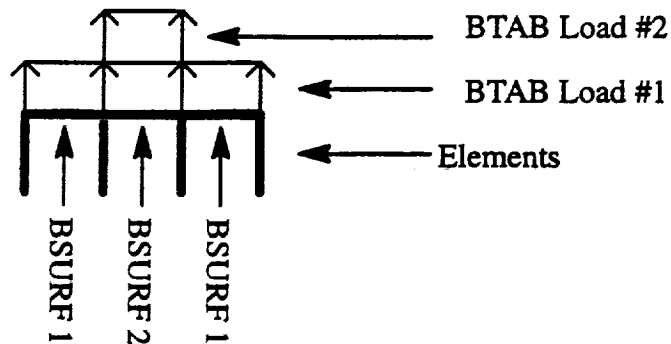
The quantity stored in the blank common vector is either the PATRAN load ID (BTAB NAME) or a packed quantity which indicates the location in a storage vector of the PATRAN load IDs that are combined on the element face. If the quantity is negative, the absolute value is the PATRAN load ID and the load is stored as a BTAB with the same NAME as the PATRAN load ID. If the quantity is positive, it is a packed sequence of vector locations. The packing is done by multiplying the previous value by 10 and adding the new vector location containing the PATRAN load ID to be combined with the previous loads on the element face. Up to 10 PATRAN load IDs can be used in combinations. It appears that 10 vector locations is the limit for the packed value due to machine precision. This results in a limit of 10 unique load IDs which can be used in various combinations. More than 10 combinations can be used, but only 10 individual loads can be used in the combinations.

An example may be illustrative:

LBSURF(101238)=159 indicates element 1238, face 2 has 3 BTAB loads applied to it. The 3 individual BTAB loads are stored with BTAB NAMES as found in LID(1), LID(5), and LID(9).

Once all the loads and load cases have been processed, the individual BTAB loads are combined into a new BTAB NAME, and the new combined BTAB NAME is stored in the element face BSURF location.

Note that this method of determining BSURFs can result in non-contiguous BSURFs for certain loadings:



## CONTROL VARIABLES

CSTEM analysis control variables are accessed from the Analysis toggle on the menu bar. Several forms are accessed from the main analysis form. These include analysis controls and load case controls. Analysis controls are those variables which are read only once in an analysis. These variables generally occur in the first 3 lines of a CSTEM deck following the title line. Load case controls are variables which are read in each load case. These variables occur on the first 2 lines of load case input in a CSTEM deck. Note that the CSTEM preference creates a single load case input deck only.

Many of the analysis and load case control variables are stored in the SIESTA database using the ANLS option. There are two load case control variables which are stored into reserved SIESTA database locations. The TIME variable is stored as CTRL data in location 37 and the NEIG variable is stored as DYAN data in location 2.

In addition to variables which can be accessed by the user from the CSTEM preference, there are a number of CSTEM control variables which are hardwired to specific values, but still stored as SIESTA ANLS data. These include the variables IPAT, IEMAG, NOISE, IABSO, NFAIL, IEIGST, and NEOUT.

Some CSTEM control variables are available only as toggles (on or off), but may have multiple options available as CSTEM input deck variables. In these cases, a specific value has been chosen as the recommended default to activate the option. Such variables include IPLO, ICRO, and IDMO, which default to the initial stiffness method when toggled on in the CSTEM preference. Also IDIS, which defaults to stress reference to original configuration, and NOUT, which defaults to restart output for last load case only. Load case control variables may change from load case to load case, but only a single load case will be written to the CSTEM input deck so no option is given to change or indicate multiple choices for these variables.

A description of all the ANLS control variables is contained in Table 3. CSTEM ANLS data is indicated by the keyword CSTI and is terminated with the keyword END.

Keyworded CSTEM input can be entered by using the Keywords... button on the Load Case Controls form. This will create a form which allows the keyword input string to be entered. Selecting the Apply button or a Return will record the keyword input string on a file called KEYWORDS. The keyword input strings are not translated into the SIESTA database. This file is later read by the deck generation executable, which inserts the file in the appropriate location of the CSTEM input deck. Note that the KEYWORDS file is initialized only once in a PATRAN session when the Analysis form is initially selected. All keyword input strings recorded during that PATRAN session will be recorded into the KEYWORDS file, even if for different databases.

<b>ANALYSIS CONTROLS</b>		
<b>keyword</b>	<b>input variable</b>	<b>description</b>
IPLO	IPLO	Plasticity Option Flag
ICRO	ICRO	Creep Option Flag
IDMO	IDMO	Damage Option Flag
IDIS	IDIS	Deformation Option Flag
IHTR	IHTR	Heat Transfer Option Flag
ICLK	ICLK	Check Run Flag
NOUT	NOUT	Output File Create Flag
NRST	NRESTA	Restart Option Flag
IRST	INREST	Restart Load Case
IPAT	IPAT	PATRAN Results File Flag
IEM	IEMAG	Electromagnetic Option Flag
NOIS	NOISE	Acoustics Option Flag
ISTR	ISTRP	Element Print Flag

<b>LOAD CASE CONTROLS</b>		
<b>keyword</b>	<b>input variable</b>	<b>description</b>
NBCK	IBUCK	Buckling Analysis Flag
ICAN	ICAN	Integrated Composites Analyzer Flag
IHTM	IHT	Heat Transfer Load Case Flag
IOUT	IPOUT	Printout Options Flag
NTIM	NTIMND	Heat Transfer Structural Sol'n Timepoint
IPC	IPC	Inelastic Property Flag
IABS	IABSO	Electromagnetic Analysis Flag
FAIL	NFAIL	Failed Stiffness Flag
IEST	IEIGST	Modeshape Stress Flag
NEOT	NEOUT	Selective Element Output Flag

<b>DYNAMIC TIME INCREMENTING CONTROLS</b>		
<b>keyword</b>	<b>input variable</b>	<b>description</b>
N2M	N2M	Creep Timestep Flag
TINT	TINIT	Initial Timestep
ECMX	ECMAX	Max Effective Inelastic Strain Increment
SGMX	SIGMAX	Max Effective Stress Change
ERMX	ERMAX	Max Integration Error
DLMN	DELMIN	Min Timestep
DLML	DELMUL	Max Timestep Multiplier

TABLE 3. ANLS KEYWORDS FOR CSTEM CONTROL VARIABLES

## HEAT TRANSFER INPUT DECK

The CSTEM heat transfer input deck is generated directly from the PATRAN database rather than from the SIESTA database. However, this process is initiated and contained within the PATRAN to SIESTA translation program. Direct generation of the heat transfer input deck from the PATRAN database is done because of the restriction to isotropic thermal material property storage in the SIESTA database and the need for time dependent thermal boundary conditions for transient heat transfer inputs. The heat transfer input deck is generated only if heat transfer loads have been applied. The heat transfer deck is written to file `f34.dat` (LUDKIA).

CSTEM heat transfer material properties are input for any number of materials, allowing for different temperature specifications from material to material. Assignment of boundary conditions to nodes and elements is done using heat transfer boundary condition (HTBC) sets, where sets are defined by combinations of boundary conditions related to a particular type of thermal load (such as convection coefficient and convection environment temperature). Boundary conditions are input at a common set of time points for a transient analysis.

The heat transfer time points are determined during translation of structural boundary conditions to the SIESTA database. The existence of each type of heat transfer load is also determined during translation of structural boundary conditions to the SIESTA database. The number of time points and vectors containing the points are saved and passed into the heat transfer input deck generation routine, along with flags indicating the existence of each type of heat transfer load.

The flags indicating the existing heat transfer load types allow the heat transfer control variables to be written. The time points, as applicable, are written to the input deck next. The heat transfer material properties (mass density, conductivities, specific heat, and radiation emissivity) are then retrieved from the PATRAN database and written to the deck.

Heat transfer loads are accessed from the PATRAN database by type in the order that they occur in the CSTEM input deck so that they can be assembled and written to the input deck before processing the next heat transfer load type. This requires multiple loops (once for each load type) over the combination of PATRAN load cases in order to assemble all loads of a given type. Since the existence or absence of each load type is known, processing can be skipped for any load types which do not exist on the database. The heat transfer loads are processed in the following order: heat generation, convection, nodal heat flux, surface heat flux, prescribed temperature, and radiation.

The values for CSTEM heat transfer boundary conditions are stored by PATRAN load variable ID in vectors SET1 to SET5 (i.e. load variable 1 in SET1, load variable 2 in SET2, etc.). As the time sequence of load variable values applied to each entity (node or element) is read, they are checked against previously stored sequences in the appropriate SETn vector to determine a matching sequence number. If no match is found, a new sequence number is created and the values for the new sequence stored in the appropriate SETn vector.

Convection and radiation loads contain multiple load variable sequences in each HTBC set. The unique combinations of load variable sequences applied to entities are stored in the array, NCOMB. The HTBC set combination for the entity under consideration is checked against previous combinations stored in array NCOMB. If no matching combination is found, a new combination number is created and the new combination is stored in the NCOMB array.

Array NCOMB contains the combinations of load variable sequence numbers. The 1st subscript corresponds to the combination number, the 2nd subscript corresponds to the load variable id, and the value is the SETn location containing the sequence of boundary condition values. For example:



a sequence of load variable 2 values is stored in SET2(3) and occurs in combination with two different sequences of load variable 1 values stored in SET1(1) and SET1(2). If these are the first 2 combinations stored in NCOMB:

Combination 1: NCOMB(1,1)=1, NCOMB(1,2)=3

Combination 2: NCOMB(2,1)=2, NCOMB(2,2)=3

The combination number applied to each entity and the number of load variables in the combination are stored in blank common vector IDSETS. IDSETS is partitioned into 6 segments of length LHT corresponding to face numbers that HTBC set combinations are applied to. The matching combination number is packed with the number of load variables and stored in the proper partition in the location corresponding to the entity ID. Non surface loads are stored as if FACE=1. For example: Combination 10 consisting of 2 load variables applied to face 3 of element 309 is stored as:

$IDSETS( (3-1)*LHT+309 ) = 10*10+2.$

It is assumed that load variables are used sequentially, so that if 2 load variables are used they are load variables 1 and 2, not 1 and 3. The packed load variable count is used mainly with radiation loads to distinguish whether gaseous emissivity and absorptivity values have been specified or not.

When all loads of the desired type are found there exist HTBC set combinations applied on entity faces stored in IDSETS, combinations of load variable sequence numbers in NCOMB, and load variable sequence values in SET1 through SET5. The assignment of HTBC set to entity is written to the deck by determining short hand notation sequences (SHNs) of entities for each loaded face. Routine SHN searches for common combination numbers for a sequence of entity IDs.

The SHN process requires 3 partitions of blank common, one of which are the combination numbers for a given face. Routine GENSHN manages the shifting and packing of partitions to free the space required for the SHN process. SHNs for each face are generated and written in turn, then all stored combinations are written by obtaining the sequence number for combination I from NCOMB(I,n) and the sequence values from SETn.

When a load type has been written, all arrays are re-initialized and the process is repeated for the next load type.

## **STRUCTURAL INPUT DECK**

The structural input deck is generated by a separate program, RDB2CSD, executed from within the PATRAN to SIESTA database translation program, pdb2deck.x. The deck generation program writes the CSTEM structural input deck from the data contained in the SIESTA database to file f35.dat (LUDKO). The deck generation program requires certain inputs, which are contained in a separate file when run by the CSTEM preference. This file is called pdb\_to\_cstem\_input and is generated by the preference based on options selected by the user from within PATRAN. The preference writes the file pdb\_to\_cstem\_input, then executes the script wrdeck. The script wrdeck begins execution of the database translation program, pdb2deck.x, indicating that input is to be taken from pdb\_to\_cstem\_input. Since it is a separate executable, the deck generation program can be run as a standalone program. The inputs must then be supplied by the user at the terminal.

## **CSTEM DECK GENERATOR**

The CSTEM deck generator program, RDB2CSD, generates the CSTEM structural input deck using the information stored in the SIESTA database, supplemented with external files containing information not available from the SIESTA database.

Some capabilities of CSTEM are not able to be utilized by the deck generation program. These are reference node transformation specifications, additional stiffnesses, ICAN material property specifications, element addition/removal, failed stiffnesses, and ICAN analysis inputs. Additionally, some parameters are hardwired to certain values. These are the geometry flag (INGEOM=0: geometry on input deck), nodal bandwidth optimization (NBAND=1: on), solution block sizing (IBLK=0: use all blank common), large displacement iteration limit (ITCVG=10), large displacement convergence tolerance (CVGTOL=.001), large displacement order (LDORDR=1), and incompatible modes indicator (INCOMP=0: include). Some options may be handled by the deck generation program, but may not be utilized by the PATRAN to SIESTA translation. These include Euler skew transformations and electromagnetic or acoustic analysis specifications.

All transformations stored in the SIESTA database are written to the CSTEM deck whether they are used explicitly in the model or not. This results in all underlying reference transformations used with PATRAN cylindrical coordinate system specifications being written to the CSTEM input deck. Also, transformations are written separately when used for nodal skews, which may result in multiple specifications of identical transformations.

CSTEM contains some keyword activated inputs including eigen solution parameters, material selected stress summaries, displacement boundary condition modifications for frequency analysis, etc. These keyword activated inputs are not stored in the SIESTA database, but are written to a separate file called KEYWORDS. The name of a file containing the keyword activated inputs is requested by the deck generator as it is generating the input deck. Either the filename or the string "NONE" should be entered. If a filename is entered the contents of the file are copied into the appropriate location in the CSTEM input deck. If "NONE" is entered, generation of the input deck continues.

Certain layered cross section specification capabilities exist in the separate deck generation program. These capabilities have not been utilized in this application. Rather, the layering specification, including cross section assignment, is read directly from the PATRAN database and written to a separate file called layer\_input, which is included into the CSTEM input deck by the deck generation program. The name of a file containing the layering inputs is requested by the deck generator as it is generating the input deck. Either the filename or the string "NONE" should be entered. If a filename is entered the contents of the file are copied into the appropriate location in the CSTEM input deck. If "NONE" is entered, generation of the input deck continues.

Nodal coordinate transformations are written to the CSTEM input deck by looping over all nodes and checking for skew coordinate systems related to boundary conditions applied to the node and to skew coordinate systems related directly to the node. If either skew coordinate system reference is found the transformation for the node is written to the CSTEM input deck using a unique transformation number. The same coordinate transformation could be written to the input deck several times, but with a different transformation number each time. If both exist, the transformation specified for the boundary condition takes precedence over the skew coordinate system for the node.

Coordinate transformations for element skews are written after coordinate systems for nodal skews. Coordinate systems other than cartesian can be used in PATRAN as element skew coordinate systems, while only cartesian coordinate systems specifications are valid in CSTEM. For this reason, any element skew coordinate system specified using other than cartesian is converted into a cartesian coordinate system specific to the element. All stored coordinate transformations are currently written so that the ones needed by element skews will be available. This includes all transformations used for nodal skews (besides SIESTA nodal angles, which are not used by the CSTEM preference)

as well as any reference coordinate systems which are used in translating cylindrical and spherical coordinate systems.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE January 2000		3. REPORT TYPE AND DATES COVERED Final Contractor Report
4. TITLE AND SUBTITLE  CSTEM User Manual			5. FUNDING NUMBERS  WU-523-22-13-00 NAS3-26617	
6. AUTHOR(S)  M. Hartle and R.L. McKnight				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)  General Electric Aircraft Engines Evandale, Ohio 45215			8. PERFORMING ORGANIZATION REPORT NUMBER  E-11858	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)  National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191			10. SPONSORING/MONITORING AGENCY REPORT NUMBER  NASA CR-2000-209308	
11. SUPPLEMENTARY NOTES  Project Manager, Christos C. Chamis, Research and Technology Directorate, NASA Glenn Research Center, organization code 5000, (216) 433-3252.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT  Unclassified - Unlimited Subject Category: 39  This publication is available from the NASA Center for AeroSpace Information, (301) 621-0390.			12b. DISTRIBUTION CODE  Distribution: Nonstandard	
13. ABSTRACT (Maximum 200 words)  This manual is a combination of a user manual, theory manual, and programmer manual. The reader is assumed to have some previous exposure to the finite element method. This manual is written with the idea that the CSTEM (Coupled Structural Thermal Electromagnetic-Computer Code) user needs to have a basic understanding of what the code is actually doing in order to properly use the code. For that reason, the underlying theory and methods used in the code are described to a basic level of detail. The manual gives an overview of the CSTEM code: how the code came into existence, a basic description of what the code does, and the order in which it happens (a flowchart). Appendices provide a listing and very brief description of every file used by the CSTEM code, including the type of file it is, what routine regularly accesses the file, and what routine opens the file, as well as special features included in CSTEM.				
14. SUBJECT TERMS  Coupled multidiscipline-structural thermal; Acoustic; Electromagnetic composites; Optimization			15. NUMBER OF PAGES 456	
			16. PRICE CODE A20	
17. SECURITY CLASSIFICATION OF REPORT  Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE  Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT  Unclassified	20. LIMITATION OF ABSTRACT	